

# A Test Chamber for the Assessment of Human Exposure to Fine Particles

M. Derudi, M. Pontiggia, G. Nano, R. Rota

Politecnico di Milano, Dip. di Chimica, Materiali e Ingegneria Chimica "G. Natta",  
via Mancinelli 7, 20131-Milano, Italy

An exposure chamber is an effective device that allows determining with precision the whole-body contact with an air contaminant, giving the possibility to effectively assess the human exposure to different toxic compounds. In this work the design of a vertical laminar exposure chamber (of about 18 m<sup>3</sup>) for aerosol contaminants is described. In order to determine the velocity and concentration fields, a computational fluid dynamics approach (CFD) has been adopted. The proposed CFD model gives an effective method for sizing rooms constituted by a distribution device which works in turbulent conditions, and an exposure room volume, provided or not by complex furnishings, working in laminar conditions. In particular, two types of aerosol have been considered: 1 µm spherical particles, which practically act as gaseous species, and 10 µm spherical particles, acting as solid. The transitory for reaching a steady-state concentration field, that is the minimum time required to reach a uniform particles distribution, have been evaluated for both the aerosols. Finally, preliminary experiments to evaluate the proper design of the whole-body exposure chamber have been carried out, confirming an almost uniform distribution of the aerosol within the test chamber.

## 1. Introduction

In working environments people are often exposed to gaseous and/or condensed pollutants that can be potential causes of allergies, diseases and in general health damages. In the framework of the occupational and environmental health and medicine, it is essential to evaluate the hazardous level of an air possible pollutant with regard to the assimilated dose (Chung et al., 1999; Rogers et al., 2005). Such an evaluation can be carried out by determining, for the sake of examples, a time-weighted amount of contaminants assimilated by workers through lunges and skin, that is by measuring the whole-body exposure (Liden et al, 1998). Theoretically, a proper exposition can be accurately determined if air and pollutant within the working environment are well-mixed, so the contaminant concentration results uniformly distributed. On the other hand, when this operational condition is not met, as it is usually observed in real environments, which are characterized by local releases of contaminants and by an inhomogeneous fluid-mechanics field, the determination of the exposure is difficult and characterized by a poor accuracy.

For this reason, an exposure chamber is an effective device that allows determining with precision the whole-body contact with the air contaminant. Brief descriptions of such rooms can be found in literature (Pauluhn, 2003), with reference to gaseous and

condensed pollutants exposition, respectively (Søstrand et al., 1997; Liden et al, 1998; Mosley et al., 2001). In these rooms, the contaminant concentration field is kept at high degree of uniformity by means of a strong mixing (turbulent exposure rooms) or by means of a top-to-bottom distribution device (laminar exposure rooms; Søstrand et al., 1997). When the pollutant is gaseous or, due to small dimensions, comparable to a gas (namely, allergens, large organic molecules, sawdust or flour) the design of exposure rooms is not critical; on the other hand, when inertial forces acting on contaminant solid particles are not negligible, that is the case of inorganic dense particles (metals, sand or PM<sub>10</sub>) the design of such rooms becomes complex and computationally expensive. In this work the design and the preliminary evaluation of a vertical exposure chamber for aerosol contaminants is described by means of a CFD approach. In particular, main process parameters used for the simulations are summarized in Table 1.

*Table 1: Main process parameters.*

Parameter	
Operating temperature	25°C
Operating pressure	1.024 atm
Chamber air flow rate	36 m <sup>3</sup> h <sup>-1</sup> (at 25°C)
Aerosol concentration	500 µg m <sup>-3</sup>
Aerosol density	2800 kg m <sup>-3</sup>
Range of aerosol dimensions	1÷10 µm

## 2. Whole-body exposure chamber design

In this work a whole-body dynamic exposure chamber (of about 18 m<sup>3</sup>) has been designed by a modelling approach, in particular by means of a computational fluid-dynamics (CFD) model, which represents nowadays a competitive and reliable technique with regards to pilot scale tests. With such an approach, velocity, scalars and particles concentration distributions can be obtained both in laminar and turbulent regime, both in stationary and transient conditions, allowing evaluating the operating performance of the test chamber. The design is targeted to reach a uniform aerosol concentration condition in the whole volume with a maximum spatial unbalance of 10% in 2 hours (Yakhot and Orszag, 1986).

Main dimensions of the exposure chamber are: 3.0 m length, 2.5 m width, and 2.4 m height; these lengths have been defined in order to ensure the possibility to host the necessary medical equipment and instruments within the test room, together with a maximum of three subjects at a time. The total net volume with internals is about 15.7 m<sup>3</sup>. At nominal operating conditions, the total flow rate was set at 36 m<sup>3</sup> h<sup>-1</sup>, that gives an ideal total air replacement rate of 0.45 h.

Modelling exposure chambers is a challenging task as typically, to reach rapidly uniform conditions, fluid-dynamic turbulent regime is required. This involves two main problems, that are: the velocity and concentration fields depend strongly on the inner chamber layout and the CFD model is computationally expensive. For such reasons, in this study an exposure chamber arrangement that minimizes both the internal layout to fluid-dynamics dependence and the computational resources has been adopted. A sketch of the equipment, without the representation of the aerosol feeding and exhaust air

systems, is reported in Figure 1. It is worth to be noticed that the chamber is divided into two main zones, a *plenum*, the higher one, and the effective exposure chamber, also named the *chamber* in the following.

The ceiling of the *plenum* is provided with four circular inlets of 0.1 m in diameter, located at 0.75 m and 0.625 m from vertical walls, whereas the floor of the *chamber* works as outlet. Conical deflector have been positioned immediately downward each circular inlet to improve the particles spreading. The PM<sub>x</sub> particles are preliminary mixed with the pure air stream, outside the *plenum*, creating a turbulent mixing inside the feeding ducts. As an intermediate ceiling, separating the *plenum* from the *chamber*, after preliminary simulations carried out both with PM<sub>1</sub> and PM<sub>10</sub> particles with and without conical deflectors, a perforated plate has been chosen; this plate, provided with holes of 0.001 m in diameter equally spaced, located at 0.2 m from the *plenum* ceiling, is moreover characterized by a total open area that is 0.13% of the total cross-section area of the test chamber, introducing a pressure drop between the two zones.

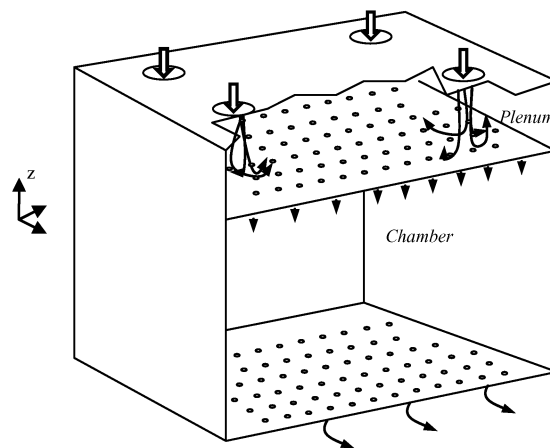


Figure 1: Sketch of the studied exposure chamber.

With such an arrangement, the *plenum* works in turbulent regime as an equalizing chamber, ensuring a uniform aerosol concentration at the inlet of the *chamber* which, in contrast, works in laminar conditions. Moreover, several simulations have demonstrated that fluid-mechanics is independent with regard to the internal layout, which leads to an effective and reliable design.

### 3. CFD modelling

According to the chamber arrangement, the CFD study has been split into two independent cases, one for the *plenum* and the intermediate perforated plate, and the other one for the laminar exposure *chamber*. Computed velocity and concentration fields at the outlet of the upper part have been applied as inlet boundary conditions for the laminar *chamber* model, where these conditions create homogeneous and vertical laminar mixing. The air is thus exhausted from outlet perforated plates, similar to those located between *plenum* and *chamber*, at the floor level.

The presence of perforated plates at top and/or bottom of the investigated domain requires a very high local refinement of the computational grid, resulting in a large number of cells and in really time-consuming simulations, especially critical to be managed when a two phases heterogeneous system, like the PMx dispersion within an airflow, is considered. For this reason, a set of preliminary simulations, considering a gas, small particles ( $PM_{1}$ ) and large solid particle dispersion ( $PM_{10}$ ), has been carried out to evaluate the possibility to simulate the dispersion through a perforated plate as a dispersion through a porous septa, a system on which it is possible to set few boundary conditions, reducing significantly at the same time mesh complexity and computational efforts. These tests have been performed on a small portion of the exposure chamber volume, provided by a square plate of about  $7 \times 7 \text{ cm}^2$ .

Then, the effective geometrical domains have been modelled; for the sake of example, in Figure 2 are reported the modelled domains for the whole chamber and for the internals for an hypothetical exposure session. The numerical technique adopted for the fluid-mechanics simulation is based on the finite volume method, consequently the geometrical domain has been converted into a computational domain by a meshing operation.

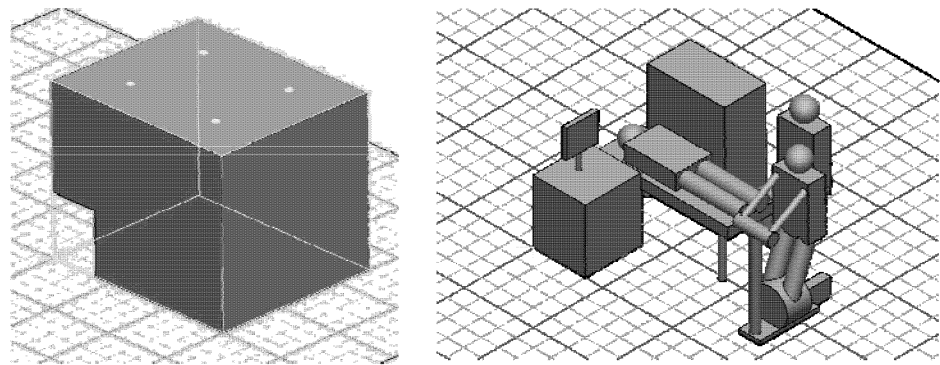


Figure 2: Geometrical domains for the whole chamber (left) and for internals (right).

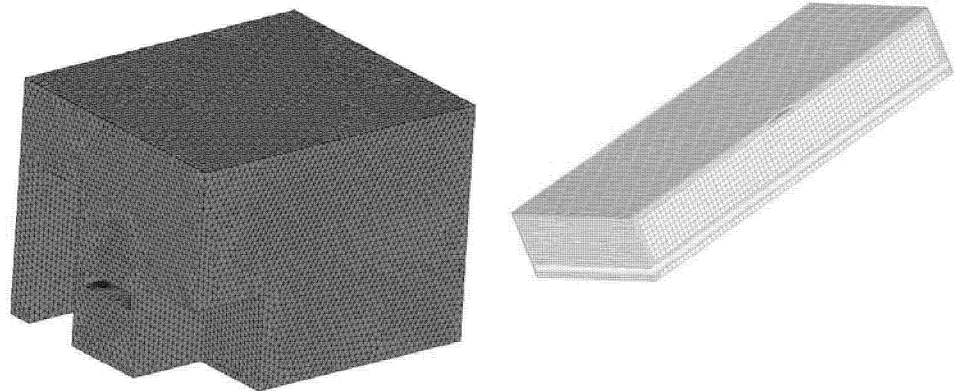


Figure 3: Computational domain for the chamber (left, unstructured mesh) and for the plenum (right, structured mesh).

Concerning the *plenum*, for which only a quarter has been considered due to symmetry, a structured mesh has been used, giving a model with 251k cells. On the contrary, an unstructured mesh has been adopted for the laminar *chamber*, made by 700k tetrahedral cells. Two details of the mesh are shown, for the sake of example, in Figure 3.

In this study the air fed to the chamber has been considered incompressible and with constant physical properties; further, since the particles concentration is small the gas-to-particle coupling was of one-way type. CFD simulations have been carried out for two different cases, that is with 1  $\mu\text{m}$  and 10  $\mu\text{m}$  solid inorganic particles. In the first case, due to the small Stokes number, the particles can be practically assumed as gaseous species; on the contrary, for heaviest particles a discrete trajectory approach has been necessarily accounted for. Governing equations for the case of small particles are mass continuity, Navier-Stokes and scalar concentration equations in transient form:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (1)$$

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} + \nabla P = \rho \mathbf{g} + \nabla \cdot \left[ \mu (\nabla \mathbf{u} + \nabla \mathbf{u}^T) - \frac{2}{3} \mu \mathbf{I} \nabla \cdot \mathbf{u} \right] \quad (2)$$

$$\frac{\partial (\rho Y_i)}{\partial t} + \rho \mathbf{u} \cdot \nabla Y_i = -\nabla \cdot (\rho D_i \nabla Y_i) \quad (3)$$

where  $\mathbf{u}$  and  $\mathbf{g}$  represent the velocity and gravity vectors,  $P$ ,  $\mu$  and  $\rho$  are the pressure, viscosity and density of the air, while  $Y_i$  and  $D_i$  are the mass fraction and mass diffusion coefficient of the aerosol, respectively; subscript “T” and  $\mathbf{I}$  identify the transposed matrix and the unit matrix. In order to simulate the behaviour of the heaviest aerosol, particles are considered of spherical shape, with constant physical properties, uniformly distributed on the inlet surfaces. Therefore, the governing equation is the momentum balance on the particle:

$$m \frac{\partial \mathbf{u}}{\partial t} = \mathbf{F}_D + \mathbf{F}_T \quad (4)$$

where  $m$  is the particle mass and  $\mathbf{F}_D$  and  $\mathbf{F}_T$  represent the drag force and the turbulent dispersion force respectively. The last term is applied for the *plenum* only, where turbulent flow occurs. In order to properly size the transitory, particles colliding on the walls has been considered lost.

Then, turbulence is modelled by the  $k$ - $\epsilon$  model, which represents a good compromise between accuracy and computational resources for complex flow fields. Accordingly, conserving equations for turbulence kinetic energy and turbulence dissipation rate must be coupled to the above equations. The maximum computed velocity was around 0.6 m/s, at the inlets. The model was implemented by means of a commercial CFD code.

#### 4. Results and discussion

On the basis of the proposed design, initial simulations have been performed for tuning the optimal arrangement of the *plenum*, that is the minimum necessary height and the

best number and location of inlets for the aerosol distribution. First of all, as previously mentioned, the possibility to simulate the perforated plate for the distribution of the aerosol within the *test chamber* with a porous septa has been verified; this assumption can be reasonably used as confirmed by Figure 4, where the concentration profiles of the particles at two distances (0.01 and 0.03 m) downstream the perforated plate and the porous septa are compared. It is possible to notice that similar concentration profiles are reached only few centimetres downstream the chamber ceiling, foreseeing no differences in the aerosol distribution within the *test chamber*.

Once the more effecting geometry has been defined, a full simulation, starting from a clean chamber up to steady-state conditions, has been performed. Figure 5, for the sake of example, summarizes the velocity magnitude (m/s) of the feed and the particles trajectories into the *plenum* for the  $PM_{10}$  case. As expected, smaller particles are more entrained by the air flow rate; on the contrary, a larger amount of heavier particles are lost on the walls. The  $1\ \mu m$  particles that reach the *chamber* are about 44% of the total particles, whereas the same fraction drops to 33% for the  $10\ \mu m$  particles. These fractions refer to the initial part of the experimental test, when the test chamber is practically clean, while they increase progressively until the steady-state conditions within the *plenum* are reached. In the worst case (heavy particles), uniform conditions within the *plenum* are reached in about 9 minutes.

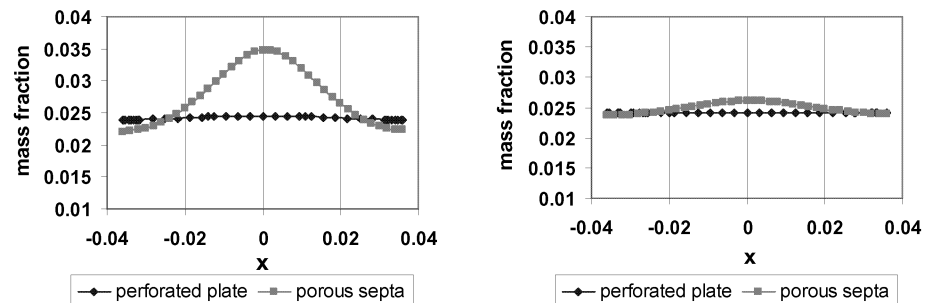


Figure 4: Concentration profiles of the particles at two distances 0.01 m (left) and 0.03 m (right) downstream the chamber ceiling for the investigated devices.

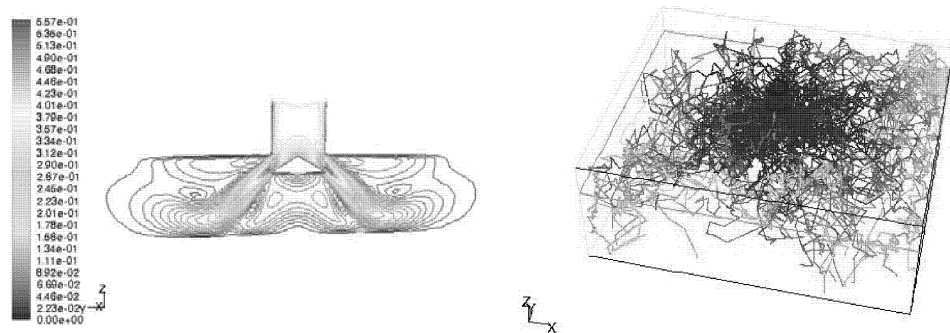


Figure 5: velocity magnitude (m/s) of the feed at the plenum inlet (left) and particles trajectories into the plenum (right) for the  $PM_{10}$  case.

The velocity and concentration fields obtained at the *plenum* outlet in steady conditions have been imposed on the upper horizontal plane of the *chamber* as inlet boundary conditions. It was found that, as worst case (light particles), the necessary transitory that allows for obtaining a uniform aerosol concentration within the *chamber* is about 2040 s. In Figure 6 concentration fields for three representative times, estimated at three locations, are reported. Clearly, the fluid-mechanics regime is laminar; however, the molecular transport is negligible with regards to the convective transport as obstacles are not promptly surrounded by the aerosol. The total transitory, for reaching sufficient uniform distribution in the *chamber*, was estimated to be 2580 s, which is considered satisfactory from a process point of view.

In order to check the quality of the numerical procedure, simulations have been performed also with a larger cells number, that is 1.5 times the described model. Grid convergence was checked by Richardson's technique (Roache, 1997); a minimum convergence of  $10^{-4}$  was obtained for each degree of freedom.

In order to check the proper design of the whole-body exposure chamber, the actual airborne dust concentration, airborne particle size, but also the behaviour of the system during transient conditions, namely during both a test start-up or a clean-up phase, will be soon evaluated by means of direct measures inside the experimental chamber but also collecting the particles onto several impactors placed within the chamber. This could be done, for example, using multi-stage cascade impactor devices, provided by membrane filters for the particles deposition (Sioutas et al., 1995).

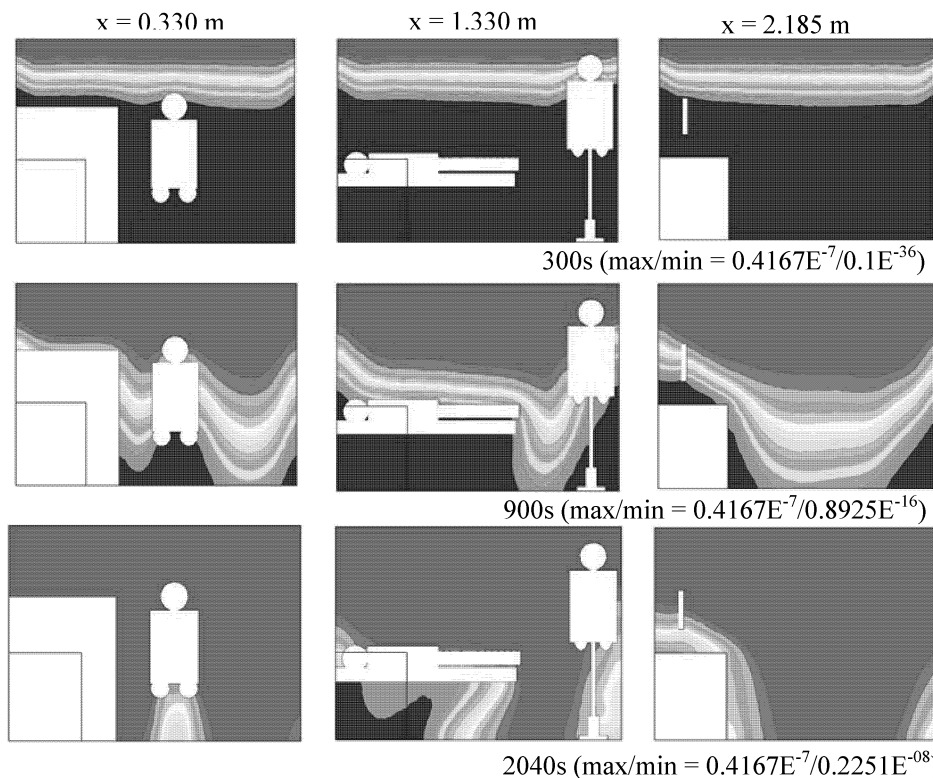


Figure 6: Aerosol concentration fields at three different locations within the chamber.

These devices, provided by a suction pump, collect at a constant flow rate the aerosol and the particles are separated from the main air flow because they impinge on different membrane filters (usually PTFE filters) located within the sampler, allowing for a rough separation of several particle size fractions and for the determination of the mass of the solids deposited on the filters surface. Such filters and impactor plates, before and after sampling, are conditioned in a controlled chamber at constant humidity and temperature for at least 24 h, then they are weighed on a micro weighing balance in order to measure the collected amount of dust. Therefore, it is possible to estimate the concentration reached by the aerosol in different positions within the chamber in order to demonstrate that spatially (location-dependent) and temporally (time-dependent) stable exposure conditions can be obtained.

## 5. Conclusions

In this work the design of a dynamic human whole-body exposure chamber for test with aerosols has been investigated and the performance of the test chamber, in terms of spatial and temporal PM<sub>x</sub> distribution and uniformity, has been preliminary assessed. Particularly, a test chamber constituted by two main parts, a turbulent air/dust mixer and sparger and a laminar exposure chamber, has been realized; a CFD approach to study the behavior of different particles size, PM<sub>1</sub> and PM<sub>10</sub>, inside the air stream and to design the chamber elements has been satisfactorily used.

Once the performance of the whole-body exposure chamber, with different materials and PM<sub>x</sub> fractions, will be completed, the main purpose of the experimental apparatus will be to perform carefully controlled exposure sessions to be used in medical studies, while an additional aim could be the possibility to develop new sampling and measuring techniques for the aerosol airborne concentration in working areas or occupational health studies.

## 6. Acknowledgments

The authors acknowledge financial support by MIUR under PRIN projects (2005).

## 7. References

- Chung C.-W., M.T. Morandi, T.H. Stock and M. Afshar, 1999, *Environ. Sci. Technol.*, 33, 3661.
- Liden C., L. Lundgren, L. Skare, G. Liden, G. Tornling and S. Krantz, 1998, *Ann. Occup. Hyg.*, 42, 541.
- Mosley R.B., D.J. Greenwell, L.E. Sparks, Z. Guo, W.G. Tucker, R. Fortmann and C. Whitfield, 2001, *Aerosol Sci. Technol.*, 34, 127.
- Pauluhn J., 2003, *Toxicology Letters*, 140, 183.
- Roache P.J., 1997, *Annu. Rev. Fluid. Mechanics*, 29, 123.
- Rogers R.E., D.A. Isola, C.-J. Jeng, A. Lefebvre and L.W. Smith, 2005, *Environ. Sci. Technol.*, 39, 7810.
- Sioutas C., P. Koutrakis, and R.M. Burton, 1995, *Environ. Health Perspect.*, 103, 172.
- Søstrand P., J. Congerud, W. Eduard, T. Nilsen, M. Skogland and J. Boe, 1997, *AIHA Journal*, 58, 521.
- Yakhot V. and S.A. Orszag, 1986, *J. Scientific Computing*, 1, 1.