



**HAL**  
open science

## Step-by-step CFD validation of turbulent particle transport and deposition in industrial bends

Jeanne Malet, Rémy Ploix, Evelyne Géhin

► **To cite this version:**

Jeanne Malet, Rémy Ploix, Evelyne Géhin. Step-by-step CFD validation of turbulent particle transport and deposition in industrial bends. FMC 2024 - XXVI Fluid Mechanics Conference, Sep 2024, Varsovie, Poland. hal-04875135

**HAL Id: hal-04875135**

**<https://hal.science/hal-04875135v1>**

Submitted on 8 Jan 2025

**HAL** is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.



Distributed under a Creative Commons Attribution - NonCommercial - NoDerivatives 4.0 International License

## Step-by-step CFD validation of turbulent particle transport and deposition in industrial bends

J. Malet<sup>1</sup>, R. Ploix<sup>1,2</sup>, E. Gehin<sup>2</sup>

<sup>1</sup> Institut de Radioprotection et de Sûreté Nucléaire (IRSN), PSN-RES/SCA/LEMAC, F-91400 Saclay, France

<sup>2</sup> CERTES . UPEC, University Paris Est, Créteil, France

E-mail: [jeanne.malet@irsn.fr](mailto:jeanne.malet@irsn.fr)

**Abstract.** Numerical tools are necessary to assess particle transport in ventilation ducts, a topic of many interests for sanitary, safety and security reasons. The objective of this study is to evaluate CFD modelling in industrial rectangular bend ducts. Sensitivity studies are performed at small-scale and the resulting observations are applied at large industrial scale focusing on deposition in bends for which experimental data are obtained on a large-scale facility especially developed for this purpose.

**Keywords:** Computational Fluid Dynamics, Particle, Aerosol, Deposition, Duct Flow, Ventilation Ducts, Experimental Fluid Mechanics, Multi-phase Flows, Turbulence

### 1. Introduction

Indoor air as well as many industrial applications imply the use of ventilation networks in which particle transport is a subject of interests for sanitary, safety and security reasons. Numerical tools are necessary to evaluate the aerosol transport and deposition under such industrial channel flows and these ones must be easy to use for applied engineering. One way to produce such numerical tools is to perform parametric CFD calculations to obtain database or develop simple correlations. However, particle transport and deposition modelled by CFD especially in bends exhibit a large range of variation and questions [1]. The objective of this study is to evaluate CFD modelling of particle transport and deposition in large scale bends of different shapes.

### 2. Experimental facility and measurement techniques

The experimental facility is a 60 m length channel of rectangular cross section with many different singularities such as horizontal and vertical bends of different shapes, reductions of different shapes and T-junctions [2]. The main channel is 400×600 mm<sup>2</sup> and the smaller one of 200×300 mm<sup>2</sup> both of galvanized steel, which makes, to our knowledge, one of the largest scale for such studies compared to previous ones [3]. This facility works under aspiration with an air conditioning station: the air is filtered; humidity is condensed, and the air is reheated to ensure a desire temperature. A honeycomb is located downstream the air conditioning section. Reynolds bulk numbers  $Re_b$  are about  $10^5$ , Reynolds numbers based on the friction velocity  $Re_\tau$  are about 9000. Aerosol is generated by a specific vibrating ceramic system that allows to

produce almost monodisperse NaCl particles between 2 and 6  $\mu\text{m}$  diameter ( $\sigma_g = 1.4$ ). The aerosol is marked with a fluorescent tracer to allow its detection on walls. High frequency hot wires (Dantec HFHW) up to  $y^+ = 5$  and PIV (LaVision) at large scale (50 cm window) have been used to characterize the flow. Particle size distribution is measured continuously with an APS (TSI). The bulk aerosol is measured by sampling on filter and a specific protocol has been developed for the measurement of aerosol deposited mass on local  $100\text{ cm}^2$  zone, without any subtract on the duct surface [4][1]. Data reduction is avoided to compare deposited mass without further modelling that might bring uncertainties (friction velocity, bulk concentration). Deposition is performed between thirty minutes and two hours. Specific attention is given to the boundary conditions derived from experiments. The size distribution is controlled during the whole steady-state and the flowrate, temperature and humidity are sampled at 1 Hz. The channel walls are insulated with 5 cm layer material. The presentation will describe the experimental facilities and validation.

### 3. Numerical study and preliminary validations

The use of an industrial scale experiment and a commercial software (Ansys Fluent v. 2022 R1) led to several questions on the code-experiment validation. Before considering the results on the large-scale duct, parametric sensitivity studies are performed at a small-scale test [5]. Since the goal here is to model bends and T-junctions flows, adverse pressure gradients make the use of wall laws inadequate. The wall mesh is thus refined to  $y^+ = 1$  in most of the domain with up to 20 layers below  $y^+ = 30$  and at least 4 nodes up to  $y^+ = 5$ . For the particle modelling, for the same reason, no wall functions are used since they are not accurate in such flows. A lagrangian stochastic particle tracking (one-way coupling) is performed considering no bouncing on walls. SIMPLE scheme is used with Green-Gauss Node-Based discretization and second order discretization. Neither customized wall-correction nor Eddy Interaction Model modifications [6] are used and Best Practice Guidelines recommended by Fluent [7] are applied.

A sensitivity study to the bulk mesh density, the type of mesh (tetra/hexa), the turbulence model (SST-k- $\omega$  and RSM based on epsilon), and the lagrangian stochastic model is performed on the small-case test and will be detailed in the presentation, illustrating the importance of these parameters to get accurate results. Calculations on the large-scale facility are then performed on the same basis of modelling choices. A flow analysis based on mean variables down to second order moments of the wall shear stresses is presented in a companion paper [8].

### 4. Code-experiment comparison and discussion

Focus is mainly given to the bend flows. Experimental results of streamwise velocity and particle mean concentration horizontal profiles upstream and downstream of a horizontal bend are presented. Aerosol mass deposited on local surfaces are also measured. These measurements are performed for three various bend shapes (smooth to sharp  $90^\circ$  angle) and are analysed using the numerical data. Aerosol deposition is mainly observed on the bend horizontal walls whereas side walls deposition is about 10 to 100 times lower.

### References

- [1] Abbasi S., A. Mehdizadeh, 2024, International Journal of Multiphase Flow **174**, 104754
- [2] Malet J., *et al.* 2022, Building and Environement, **222**, 109223
- [3] Sippola M. R., W. W. Nazaroff, 2004 *Aerosol Science and Technology*, **38** 9
- [4] Costa D., J. Malet, E. Géhin 2022 *Meas. Sci. Technol.* **33** 9
- [5] Pui D. Y., H. F. Romay-Novas, B. Y. Liu., 1987, *Aerosol Science and Technology* **7:3** 301
- [6] Jubaer H., *et al.*, 2024, *J. Aerosol Sci.* , **175** 106262
- [7] Menter F.R., *et al.*, 2021, BPG: RANS Turbulence Modeling in Ansys CFD, Ansys Germany GmbH
- [8] Ploix P., J. Malet, E. Gehin, 2024 *Proc. XXVI Fluid Mechnaics Conference*, (Warszaw, Poland)