HARMO22

22nd International Conference on
Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes
10-14 June 2024, Pärnu, Estonia

APPLICATION OF THE BUILD OPERATIONAL DISPERSION MODEL FOR ACCIDENTAL OR DELIBERATE RELEASES IN REAL COMPLEX URBAN AND INDUSTRIAL AREA

Lionel Soulhac[1]*, Guillevic Lamaison*[2]*, Perrine Charvolin*[2]*, Chi Vuong Nguyen*[2]*, and Patrick Armand*[3]

[1] INSA Lyon, CNRS, Ecole Centrale de Lyon, Universite Claude Bernard Lyon 1, LMFA, UMR5509,
69621, Villeurbanne, France
[2] Ecole Centrale de Lyon, CNRS, Universite Claude Bernard Lyon 1, INSA Lyon, LMFA, UMR5509,
69130, Ecully, France
[3] CEA, DAM, DIF, F-91297 Arpajon, France

Abstract: The Building Urban & Industrial Lagrangian Dispersion (BUILD) model is an operational atmospheric dispersion model developed for accidental or deliberate release in urban or industrial area. The BUILD model is based on an improvement of a “SIRANE like” street network parameterization for the dense part of an urban area associated with an obstacle-wake parameterization to describe the flow around and downwind of buildings. A stochastic Lagrangian particles approach is coupled with the parameterization of the flow to simulate the transport and dispersion of pollutants in the domain. Different kind of sources can be considered and specific processes like plume rise, deposition, radioactive decay, micro-mixing model for the fluctuations and dose calculation are considered. The BUILD model has been optimized, with parallelization and domain splitting, in order to simulate the dispersion in a complex building’s arrangement in a computational time less than few minutes on a laptop. The present paper describes some improvements of the flow parameterizations and some results on idealized or more realistic application cases.

Key words: *Atmospheric dispersion modelling, Lagrangian model, emergency response,*

INTRODUCTION

Accidental or intentional release of chemical, biological, radiological, or nuclear (CBRN) materials is a significant threat to public safety and infrastructure security, especially in densely populated urban and industrial areas. To effectively respond to such emergencies, first responders, authorities, and industrial companies need operational simulation tools to take decisions and support actions.

To address this need, LMFA and CEA have developed the BUILD (Building Urban and Industrial Lagrangian Dispersion) model (Soulhac et al., 2022). The BUILD model simulates the transport of CBRN pollutants within complex built environments. It leverages an improved parameterization based on the street network approach for dense urban areas (Soulhac et al., 2011, 2012, 2017) alongside an obstacle-wake parameterization to capture airflow patterns around buildings. The model adopts a stochastic Lagrangian particle approach within this simplified flow field to simulate pollutant transport and dispersion. It also accounts for specific processes like plume rise, deposition, radioactive decay and concentration fluctuations (see paper H22-146, Soulhac et al., 2024). BUILD has been optimized for speed and efficiency through parallelization and domain splitting techniques.

This paper focuses on the parameterisation proposed to describe flow and dispersion around 3D obstacles. In the next section, we firstly provide an overview of the BUILD model. Then we present in more details the original parameterization for the flow around obstacles, dedicated to area of intermediate or low buildings density. Finally, the last section presents some application cases on idealized of more realistic urban and industrial configurations, in order to highlight the performance of the model and its limitations, suggesting further developments.

OVERVIEW OF THE BUILD MODEL

The BUILD model has been developped for simulating flow and atmospheric dispersion at local scale, assuming a simplified representation of the influence of buildings and obstacles, particularly in urban or industrial environments. It is a rapid response operational software, based on approximated parameterizations of the main flow features, developped as a compromise between physical realism of the results and numerical performance for operational use. The model aims to provide concentration maps of the evolution of a cloud of dangerous materials of the CBRN type, in the context of industrial or chemical accidents and of malicious acts.

The BUILD model assumes a simplified description of the buildings topography of the area of interest. The domain is divided into 1 km x 1 km tiles. Each tile is described with a resolution of 1 meter. Using a set of Matlab Image Processing Toolbox algorithms, the geometry of the buildings is simplified in order to identify the main topological characteristics of the canopy. More specifically, the geometrical preprocessor calculates the structure of the network of interconnected street canyons and their geometrical parameters (width and height). It also precalculates several flow patterns around obstacles (see next section for details).

The BUILD software is based on parameterized modeling of the flow in the atmospheric boundary layer and in the obstacle canopy, with a spatial decomposition of the study domain into four types of zones, as illustrated on **Figure 1** and described below:

- Atmospheric boundary layer:
  - Mixed and inertial atmospheric boundary layer: in this zone, BUILD adopts classical parametric Monin Obukhov similarity profiles for the mean and turbulent characteristics of the flow.
  - Roughness sublayer: in this zone, the model modifies the parametric boundary layer profile with a simplified flow model to take into account the disturbance induced by obstacles (see next section for details).
- Canopy of obstacles:
  - Dense urban environment: an analytical model of the flow in a network of street canyons, connected by street intersections, is used. This is an evolution of the flow model adopted in the SIRANE and SIRANERISK software (Soulhac et al., 2011, 2022).
  - Intermediate or low buildings density area: in this zone, a new semi-analytical model is proposed to describe the flow in the recirculation zones upwind and downwind of obstacles (see next section for details).

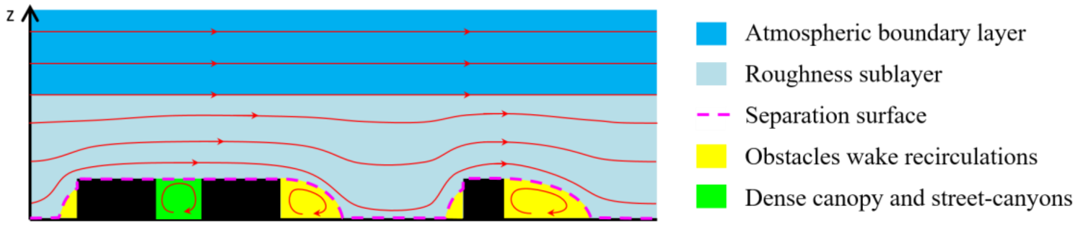


Figure 1. Diagram of the different flow zones used in the BUILD model.

The BUILD model parameterization assumes the existence of a separation surface (in dashed purple on **Figure 1**) between the canopy zone (obstacles wake recirculations and dense canopy) and the overlying boundary layer (roughness sublayer and mixed / inertial atmospheric boundary layer). Over the separation surface, the flow can be deviated and accelerated according to the constriction of the roughness boundary layer but we assume that no back flow can occur. Below the separation surface, back flows and recirculations are the main features of the velocity field.

A stochastic Lagrangian particle model is finally used for the simulation of pollutant dispersion, considering the mean velocity and turbulence fields described previously to transport and disperse particles. The evolution of the fluctuating velocity U_i' is determined by a stochastic differential equation (Thomson, 1987):

$$dU_i' = a_i(\boldsymbol{X}, \boldsymbol{U'}, t)dt + \sum_j b_j(\boldsymbol{X}, \boldsymbol{U'}, t)d\xi_j \qquad (1)$$

where a_i and b_j are expressed in terms of standard deviations of the velocity fluctuations σ_{u_i} and of the Lagrangian time scales $T_{L,i}$.The concentration is calculated on a grid mesh by box-counting of the particles.

In the next section, we detail the last developments of the parametric model for the flow around obstacles in intermediate or low buildings density area.

PARAMETRIC FLOW MODEL AROUND OBSTACLES

In this section, we firstly present how the BUILD model evaluates the geometry of the interface between the wake recirculation zone and the roughness sublayer zone. Then, we detail the parameterization for the flow below (in the recirculation zones) and above (in the roughness sublayer) this interface.

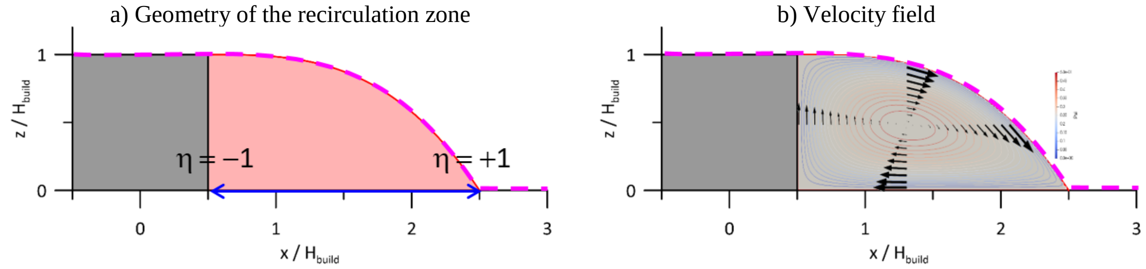
Interface between the wake recirculation zone and the roughness sublayer zone

In the close vicinity of each obstacle, the geometrical boundary of the main recirculation areas, upwind and downwind of each obstacle, is precalculated, considering that these recirculations represent the main feature of the flow around the obstacle. For this, we simulate the advection-diffusion of the velocity defect induced by the obstacle to determine the shape of the recirculation zone. This process is emulated by an image processing algorithm, using translation-blurring of the image of each building, with blurring parameters calibrated to reproduced the momentum diffusion in the wake. The ground footprint of the recirculation is estimated by thresholding the velocity defect field, adjusting the threshold to get the desired recirculation length provided by empirical rules (Hosker, 1985). This algorithm is applied downwind and also upwind of the buildings, using different threshold values to reproduce realistic characteristics of the flow topology. Application of this method for the upwind recirculation has less physical founding but is a convenient and efficient way to estimate the displacement zone.

The velocity defect field is also used to calculate the height of the top of the recirculation zones. This height, completed with the heights of the buildings and of the streets, defines the separation surface between the canopy and the roughness sublayer. Considering a normalized longitudinal coordinate η inside the recirculation ($\eta = -1$ on the building leeward side and $\eta = +1$ at the end of the recirculation), the height of the recirculation is calculated following the analytical function:

$$h_r(\eta) = H_b\left[1 - \left(\frac{1+\eta}{2}\right)^3\right] \qquad (2)$$

An example of the recirculation zone obtained by this approach downwind of an obstacle is illustrated on **Figure 2**-a


a) Geometry of the recirculation zone

b) Velocity field

Figure 2. Recirculation zone (a) and velocity field (b) downwind of an idealized obstacle.

Wake recirculation model

Once the geometry of each recirculation is known, the flow inside it is calculated assuming an analytical form of the stream function ψ:

$$\psi(\eta, \zeta) = q(1 - \eta^2)(1 - \zeta^2) \quad \text{with} \quad \zeta = \frac{2z}{h_r(\eta)} - 1 \qquad (3)$$

Where q is a constant (recirculation intensity) related to the wind velocity in the overlying boundary layer. The velocity components in a vertical plane aligned with the external wind direction are related to the stream function by:

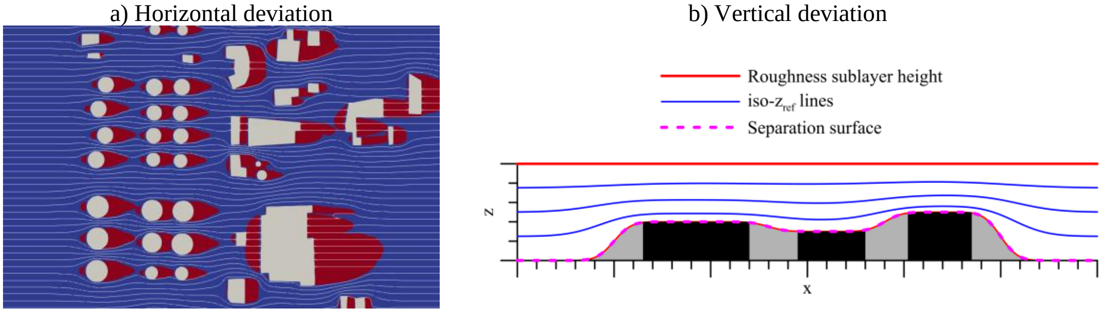
$$\bar{u} = \frac{\partial \psi}{\partial z} \quad \text{and} \quad \bar{w} = -\frac{\partial \psi}{\partial x} \qquad (4)$$

The resulting velocity field is illustrated on **Figure 2**-b.

Roughness sublayer model

The flow model inside the roughness sublayer is a parametric approach which combines two deviation effects:

- An horizontal deviation of the streamlines which flow around the obstacles due to their blocking effect (see **Figure 3**-a). This behaviour is calculated by solving a Laplace equation for the horizontal stream function, assuming that the lateral displacement around obstacles is mainly an inviscid process that can be modelled as a potential flow. For wide buildings, the model allows the flow to be advected through the footprint of the obstacle, because the flow is mainly deviated vertically in this case and passes over the obstacle.
- Vertical deviation of the streamlines which are moved upwards due to the elevation of the separation surface (see **Figure 3**-b). This effect is modelled simply by a progressive geometrical deformation of the streamlines between the separation surface at the top of the canopy and the horizontal surface at the top of the roughness sublayer.

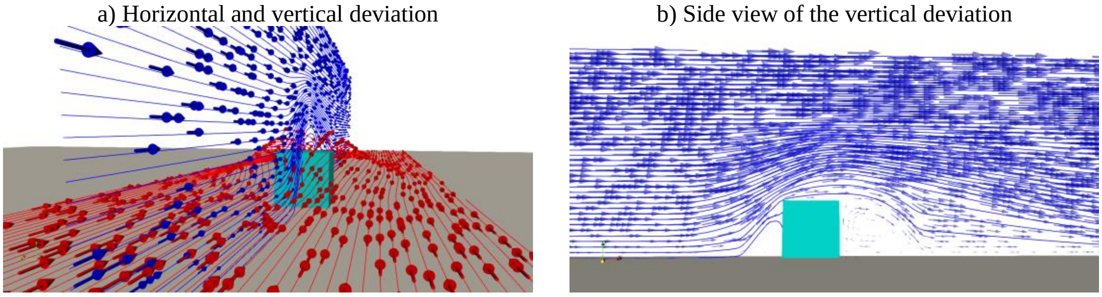


a) Horizontal deviation

b) Vertical deviation

Figure 3. Horizontal and vertical deviation of the flow in the roughness sublayer

Obviously, horizontal and vertical deviations occur simultaneously and the BUILD model merges both effects, assuming that close to the separation surface, horizontal deviation dominates while at the top of the RSL, it becomes negligible. An example of the 3D flow calculated by this methodology around a cubical surface mounted obstacle is presented in **Figure 4**.



a) Horizontal and vertical deviation

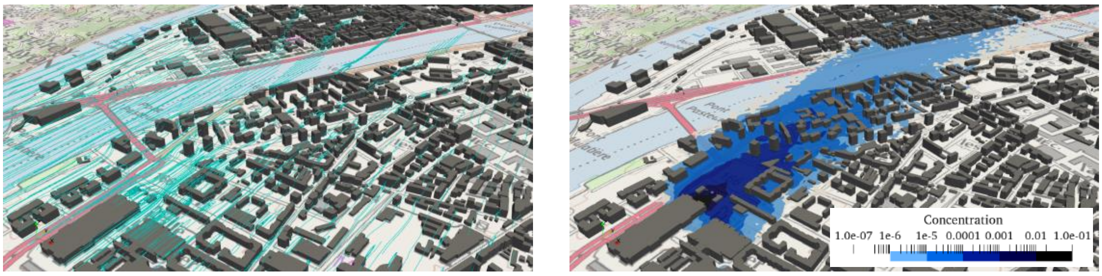
b) Side view of the vertical deviation

Figure 4. Horizontal and vertical deviation around an isolated cubical surface mounted obstacle.

APPLICATIONS OF THE BUILD MODEL

The BUILD model, modified with the new parameterization for the flow over complex 3D buildings, has been applied on different test cases. Firstly, the flow and the dispersion of pollutant emitted at a point continuous release has been simulated on some idealized test cases (isolated obstacle of different aspects ratios, group of few obstacles) in order to evaluate the model, from a physical and numerical efficiency point of view, and some sensitivity studies have been performed.

Then the model has been applied on more realistic test cases of dispersion on real urban or industrial areas (see **Figure 5** for an application in Lyon). The results demonstrate the ability of the model to reproduce the main characteristics of the pollutant plume, with a very short computational cost compared with more complex flow and dispersion models, like RANS or LES CFD approaches associated with Lagrangian dispersion models. This application also highlights the need to improve some aspect of the velocity and turbulence fields, especially at the interface between the different zones where various parameterization are applied.



Flow streamlines

Concentration field

Figure 5. Application of the BUILD model on a real district of 3km x 3km on the city of Lyon.

CONCLUSION

In this paper, a new model for flow and dispersion around complex 3D obstacles is presented in the context of the BUILD model development. The model is based on simplified flow parameterizations coupled with a Lagrangian dispersion model, with the aim to produce realistic results in short computational time, compatible with operational applications in urban or industrial areas. The model has been applied and evaluated on different test cases in order to discuss results and suggest further improvements.

REFERENCES

Hosker, R. P., 1985. Flow around isolated structure and building clusters: a review. In ASHRAE Transactions.

Soulhac, L., Perkins, R. J., and Salizzoni, P., 2008. Flow in a street canyon for any external wind direction. *Boundary-Layer Meteorology*, **126(3)**, 365-388.

Soulhac, L., Salizzoni, P., Cierco, F. X., and Perkins, R., 2011. The model SIRANE for atmospheric urban pollutant dispersion; part I, presentation of the model. *Atmospheric environment*, **45(39)**, 7379-7395.

Soulhac, L., Salizzoni, P., Mejean, P., Didier, D., and Rios, I., 2012. The model SIRANE for atmospheric urban pollutant dispersion; PART II, validation of the model on a real case study. *Atmospheric environment*, **49**, 320-337.

Soulhac, L., Nguyen, C. V., Volta, P., and Salizzoni, P., 2017. The model SIRANE for atmospheric urban pollutant dispersion. PART III: Validation against NO2 yearly concentration measurements in a large urban agglomeration. *Atmospheric environment*, **167**, 377-388.

Soulhac, L., Lamaison, G., Charvolin, P., Nguyen C. V., Slimani, M., and Armand, P., 2022. Development and validation of the BUILD operational dispersion model for accidental or deliberate releases in complex area. Harmo21. Conf., Aveiro, Portugal.

Soulhac, L., Lamaison, G., Charvolin, P., Nguyen C. V., and Armand, P., 2024. A new operational micromixing approach to model concentration fluctuations. Harmo22. Conf., Pärnu, Estonia.

Thomson, D. J., 1987. Criteria for the selection of stochastic models of particle trajectories in turbulent flows. *Journal of Fluid Mechanics*, 180, 529 56.