

A VALIDATION OF THE FIRE DYNAMICS SIMULATOR FOR SMOKE DISPERSION FROM METRO STATIONS

¹Conor Fleming, ²Norman Rhodes

¹SNC-Lavalin, Vancouver, Canada

²Independent consultant, Washington DC, USA

ABSTRACT

In the event of a fire in a metro system, exhausted smoke should be prevented from recirculating into the underground space or entering neighbouring buildings. This can be achieved through judicious positioning of smoke exhaust vents and air intakes, and the design performance can be verified through numerical modelling.

The modelling method should capture the underlying flow physics at the relevant length and time scales, so that smoke transport can be predicted to a practical degree of accuracy. In the context of metro station smoke recirculation, the region of interest encompasses local air intakes and is likely limited to one or two street canyons. The flow field at this scale is highly unsteady, comprising freestream turbulence associated with the atmospheric boundary layer as well as eddies generated by vortex-shedding from buildings.

Fire Dynamics Simulator (FDS) is expected to be a suitable modelling tool due to its large-eddy simulation formulation. The dominant eddies at building scales are simulated directly, and a simple turbulence model is applied to capture the effect of the smallest eddies. FDS has been comprehensively validated for buoyancy-driven flow and mass transport problems, and is widely used in the design and analysis of tunnel ventilation systems. In this study we use FDS version 6.5.3 to predict the flow around rectangular buildings at model scale and show that it compares favourably with experimental results and more established numerical tools (e.g. commercial RANS and LES software packages).

Keywords: Fire Dynamics Simulator, urban dispersion, metro station smoke dispersion

1. INTRODUCTION

The work described in this paper was prompted by a requirement in the design of Toronto's Eglinton Crosstown Light Rail Transit (ECLRT) line to check that smoke emitted from ventilation shafts in the event of a fire does not recirculate to an extent that compromises station entrances.

The ECLRT line will carry passengers along Eglinton Avenue from Weston Road in the west to the Kennedy TTC subway station in the east. The new rapid transit line will include a 10-kilometre underground portion between Keele Street and Laird Drive, running on dedicated right-of-way transit lanes that will be separated from regular traffic for the rest of the route to ensure reliable, fast travel times for customers.

2. METHODS

The simplest dispersion predictions employ Gaussian plume methods. They are widely used and highly developed for specific applications, but are generally more applicable to large scale problems. The unsteady, three-dimensional nature of flow in the near field of buildings is not compatible with these methods, and so the more sophisticated (and more expensive) CFD approach is adopted.

Two different CFD codes have been used in this study, one using a RANS method with a $k-\varepsilon$ turbulence model and the other using FDS with its large-eddy simulation (LES) turbulence model. The RANS method may have the advantage of economy since a steady-state solution can be obtained. However, unsteady periodic fluctuations can occur around obstacles and this may invalidate the steady state assumption. LES is inherently unsteady, and should be capable of predicting such fluctuations as long as the dominant eddy scales are resolved adequately.

The boundary conditions for a RANS simulation are fairly straightforward to implement. Vertical profiles of velocity, turbulence kinetic energy and dissipation rate can be prescribed at the inlet boundary. In an LES simulation, the process is more complex.

A fundamental challenge in LES modelling is the prescription of an unsteady, turbulent velocity profile at the upstream boundary. The conventional method for channel flows is to generate and record a suitable inflow profile via a precursor simulation which is periodic in the streamwise direction. To avoid the computational effort associated with the precursor simulation, Jarrin (2008) developed a synthetic eddy method, where eddies are generated during the simulation by flow perturbations near the inflow boundary and convected into the domain. In this model, available in FDS, the magnitudes and frequencies of these perturbations are set automatically to reflect the user-defined turbulence length scale, l , eddy density (or ‘number of eddies’), N , and root-mean-square of velocity fluctuations, u_{RMS} .

Ideally, the dominant turbulence length scale could be determined from high-frequency velocity measurements. In the absence of such data, we have chosen the building height as the input eddy length scale so that freestream turbulence has a similar length scale to building-generated turbulence.

Jarrin (2008) derives a formula for estimating the eddy density, $N = V_B/l^3$, where V_B is the volume of a three-dimensional, virtual ‘box’ enveloping the inlet boundary, within which the eddies are generated. The volume of this box is $V_B = (w + 2l)(h + 2l)(2l)$, where w and h are the width and height of the inlet boundary respectively.

The RMS velocity can be calculated from turbulence intensity, I , as $u_{\text{RMS}} = \bar{u}I$, where \bar{u} is the time-averaged velocity, or from turbulence kinetic energy, k , as $u_{\text{RMS}} = \sqrt{2k/3}$.

Note that FDS can only accommodate piecewise-constant profiles of turbulence parameters at the inlet.

3. VALIDATION CASES

Two cases have been studied to assess FDS and the more common RANS approach to dispersion simulation, and to establish a correct methodology for this type of problem. Both cases consider the flow over rectangular buildings at small scale. The first is a single building, taken from the Architectural Institute of Japan Guidebook for Practical Applications of CFD to Pedestrian Wind Environment around Buildings (AIJ, 2016; see also Tominaga et al, 2008). Figure 2 shows the geometry and the location of measurements of velocity and turbulence.

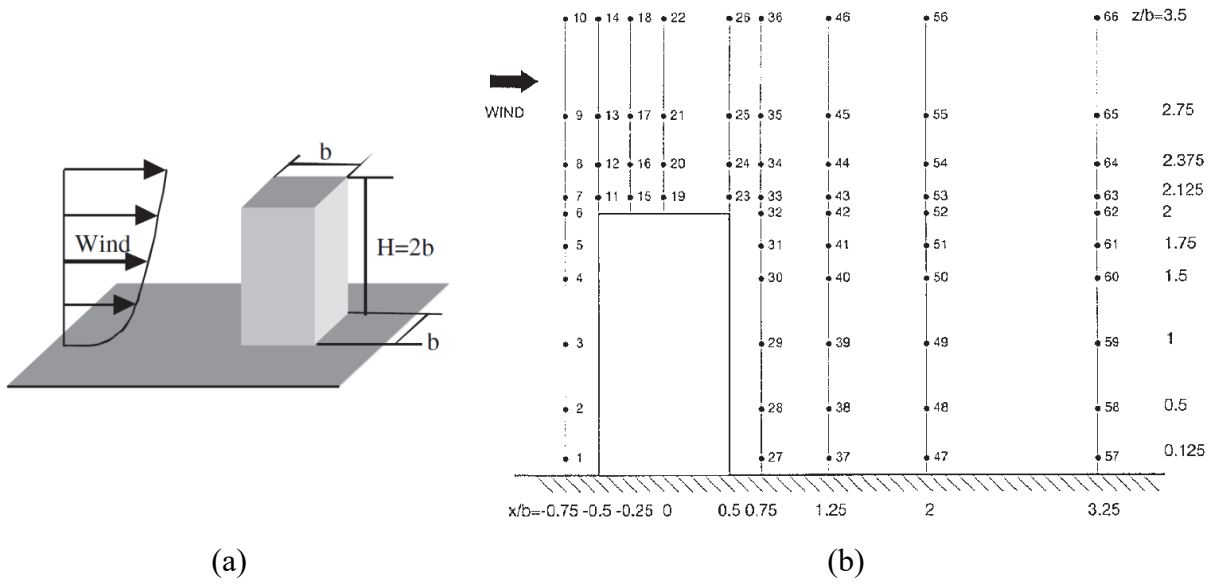


Figure 2: Illustrations of (a) geometry and (b) measurement locations in the vertical mid-plane for case 1: flow past an isolated building (AIJ, 2016).

In the second case, Chang & Meroney (2003; see also Chang, 2006) considered arrays of up to twelve buildings as shown in Figure 2. In these experiments, several ratios of building height, H , to spacing, B , were considered, as well as the number of upstream building rows, N . A tracer gas (ethane) was introduced from a point source in the centre of the array and species concentration measurements were made along the lines shown in Figure 3. The present paper considers the case $B/H = 1$ and $N = 1$. It should be noted that the inlet boundary conditions for this case were not well defined by Chang & Meroney.

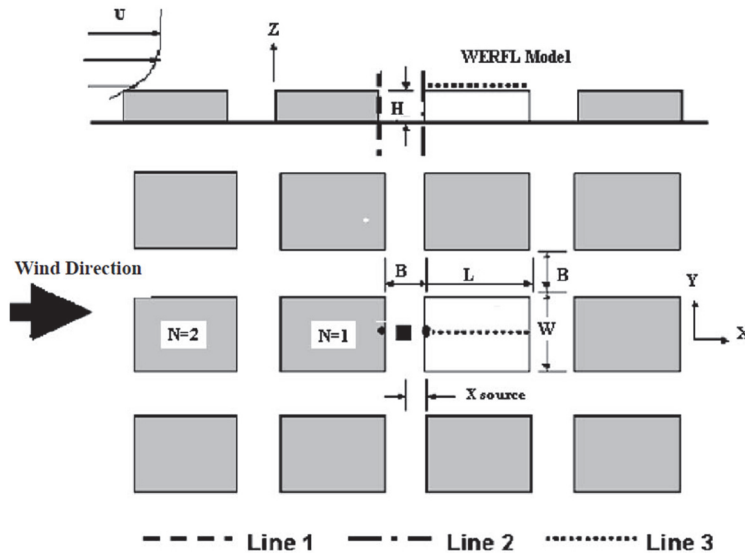


Figure 3: Illustration of geometry and measurement locations for case 2: dispersion in a street canyon (Chang, 2003). Note that the $N = 1$ case is simulated here, i.e. only a single row of buildings upstream of the source.

The geometry for both problems reflects the wind tunnel dimensions of the experiments. In both cases orthogonal grids were used and the general computational setup is similar to Gousseau et al (2013) for Case 1 and Chang & Meroney (2003) and Chang (2006) for Case 2. Velocity and approximate turbulence profiles were prescribed at the inlet boundary (the left side in Figures 2 and 3) and appropriate wall and pressure boundaries were prescribed on the other faces of the solution domain. The FDS model for Case 1 featured a uniform grid (rather

than stretched) of cubic elements with edge length $\Delta x = H/20$ in the near field, and a near-field element edge length of $\Delta x = H/16$ for Case 2. The FDS timestep was adjusted automatically to maintain a maximum Courant number of $C = ut/\Delta x_{\min} < 0.9$, where t is time.

Typical results for Case 1 are shown in Figures 4 (a) and (b) which present normalized velocity and turbulence kinetic energy profiles respectively. Three sets of results are presented, FDS and RANS simulations carried out by the authors, and for turbulence, an additional LES prediction from Gousseau et al (2013) is included. These results are for profiles on the vertical mid-plane of the building, $y/b = 0$, at the downstream location $x/b = 1.25$, where b is the building dimension in the streamwise direction and x is measured from the centre of the building.

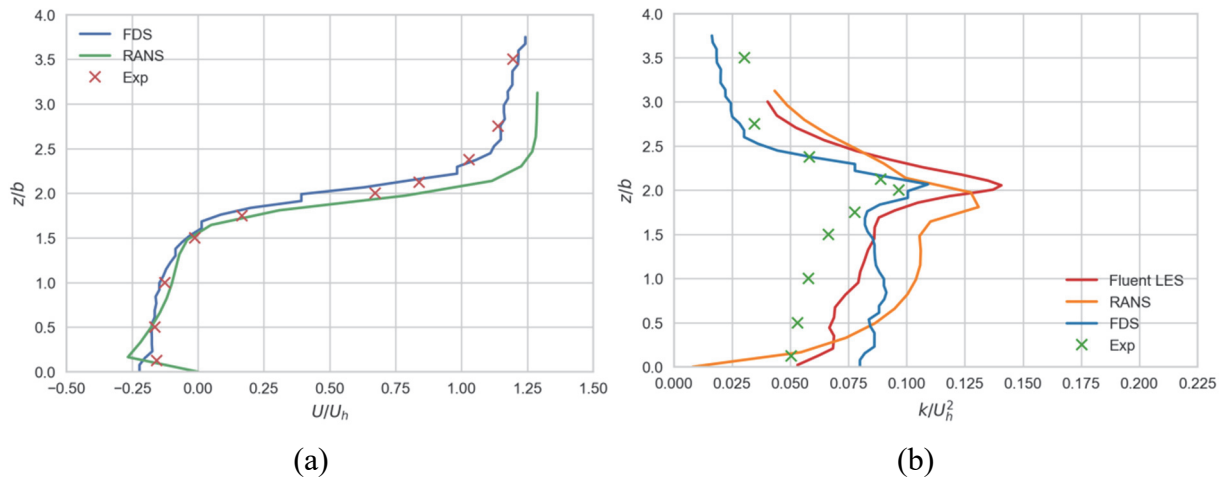


Figure 4: Normalized profiles of (a) velocity and (b) turbulence kinetic energy along a vertical line at $y/b = 0$ and $x/b = 1.25$.

It can be seen that the velocity profile at this position is well predicted by both CFD codes. All the codes somewhat over predict the turbulence kinetic energy, both the peak, which occurs at the building height ($z/b = 2$) and in the wake of the building below $z/b = 2$. In practice this over-prediction might result in greater dispersion of a scalar quantity, yielding slightly lower values of peak concentration, suggesting that the prediction of ground-level concentration might be non-conservative. Rigorous grid studies have not been carried out by the authors and so it is possible that these results might be improved by finer resolution. However, Gousseau et al reported deterioration in their LES prediction with finer grids – a rather alarming conclusion which is discussed by the authors.

The second test case proved to be more challenging, as the boundary conditions were not adequately defined – although profiles of velocity, kinetic energy and dissipation were given, the normalising velocity was not. Had the experiment been independent of Reynolds number, this would not have been important, but studies showed a strong sensitivity to the velocity magnitude.

The RANS simulations that were carried out in a steady-state mode gave results that were of the correct order, but were greatly influenced by the inlet turbulence properties. It was felt that the flow in the experiment was probably unsteady, and further work is required to elaborate this potential effect.

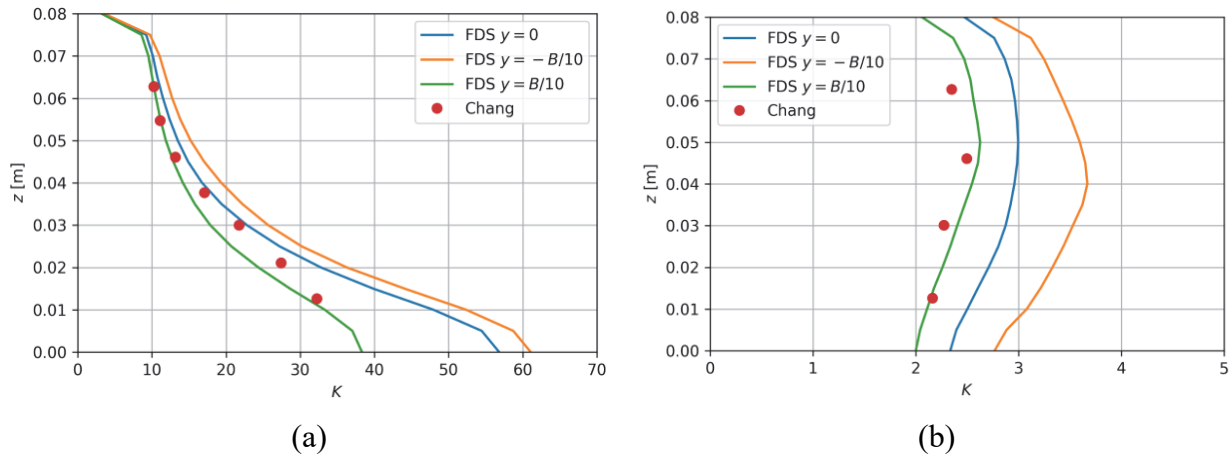


Figure 5: Vertical profiles of normalized concentration along (a) Line 1 and (b) Line 2.

The unsteady FDS simulations gave more acceptable results over a range of Reynolds numbers. Vertical time-averaged distributions of normalized tracer concentration K are presented in Figures 5 (a) and (b), where $K = \rho u_{\text{ref}} H^2 / Q$, ρ is concentration, $Q = 8.45 \times 10^{-6} \text{ m}^3/\text{s}$ is the flow rate of the tracer gas source and the averaging period is 8 seconds. The reference velocity at building height, $u_{\text{ref}} = 10.6 \text{ m/s}$, was inferred from related experiments in the same facility (Chang & Meroney, 2001; Ham & Bienkiewicz, 1998). Line 1 is on the face of the building upstream of the vent and Line 2 on the building downstream of the vent. A circulation of the flow in the cavity between the buildings causes the tracer gas to be convected toward the upstream building. Hence, the concentrations are noticeably higher at the base of the building and decrease with height (figure 5a). The concentrations on the face of the downstream building (figure 5b) are an order of magnitude smaller. In Figure 5 (a), the concentration at the centreline ($y = 0$) matches the data quite well. The concentrations on either side ($y = \pm b/10$) show an asymmetry which is consistent with unsteadiness in the flow. A similar conclusion is drawn from Figure 5 (b).

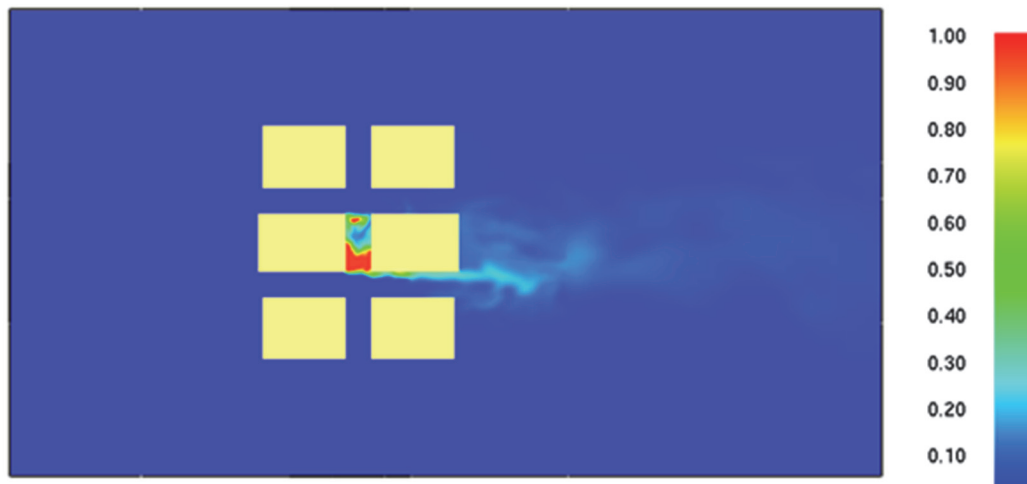


Figure 6: Instantaneous contours of tracer concentration at the horizontal mid-plane of the building array, $z = H/2$, with red contours indicating concentrations of $\rho > 10^{-3} \text{ kg/m}^3$. Note flow is from left to right and the second downstream row of buildings has not been modelled as it is expected to have little influence on near-field dispersion.

Observations of the flow field at various points in time reveal that the concentration plume tends to be biased towards one end of the street canyon, and only switches end occasionally. Figure 6 is an instantaneous contour plot of tracer concentration where the plume is biased in the negative y direction. It is interesting to note that the experiments did not indicate this

effect, and it was likely to have been averaged out by the tracer gas measuring instrumentation.

The study shows that FDS can be used to predict scalar concentrations with reasonable accuracy. Sufficient averaging time would need to be simulated to provide a complete picture since the plume clearly shows a movement from side to side and in different building arrangements this may cause excessive swings in predicted concentration. Too short an averaging time may lead to incorrect conclusions.

The RANS simulations, in a steady state mode, may not capture the intermittency of turbulence in the canyon. Further study is required using LES and unsteady RANS simulations to develop a further understanding of this effect.

4. SUMMARY

FDS shows good potential for studying the impact of smoke from metro fires, and the validation studies carried out against experiment give confidence in the full-scale predictions as well as pointers to an appropriate methodology for prescribing boundary conditions and for the critical assessment of the resulting predictions. The level of validation for the RANS simulations, although as yet incomplete, have also indicated limitations of steady state methods in this context.

The current study has focused on orthogonal flow, whereas it is generally necessary to also consider non-orthogonal winds in design simulations. In future work we intend to examine an atmospheric wind model based on Monin-Obukhov similarity theory which has recently been introduced to FDS (McGrattan et al, 2017).

5. REFERENCES

- Architectural Institute of Japan (2016) AIJ benchmarks for validation of CFD simulations applied to pedestrian wind environment around buildings. ISBN978-4-8189-5001-6, AIJ, Tokyo. URL: https://www.aij.or.jp/jpn/publish/cfdguide/index_e.htm
- Chang, C.H. Meroney, R.N. (2001) Numerical and physical modeling of bluff body flow and dispersion in urban street canyons. *J. Wind Eng. Ind. Aerodyn.*, 89, 1325-1334
- Chang, C.H. Meroney, R.N. (2003) Concentration and Flow distributions in urban street canyons: wind tunnel and computational data. *J. Wind Eng. Ind. Aerodyn.*, 91, 1141-1154
- Chang, C.H. (2006) Computational fluid dynamics simulation of concentration distributions from a point source in the urban street canyons. *J. Aerosp. Eng.* 2006.19:80-86
- Gousseau, P., Blocken, B., van Heijst. G.J.F. (2013). Quality Assessment of Large-Eddy Simulation of Wind flow around a high rise building: Validation and Solution Verification. *Computers and Fluids* 79 120-133
- Ham, H.J., Bienkiewicz, B. (1998) Wind tunnel simulation of TTU flow and building roof pressure. *J. Wind Eng. Ind. Aerodyn.*, 77&78, 119-133
- Jarrin, N. (2008) Synthetic inflow boundary conditions for the numerical simulation of turbulence. *PhD thesis, University of Manchester*
- McGrattan, K., McDermott, R., Hostikka, S., Floyd, J., Vanella, M., Weinschenk, C., Overhold, K. (2017) Fire Dynamics Simulator Technical Reference Guide, NIST Special Publication 1018-1, Sixth Edition. URL: <https://pages.nist.gov/fds-smv/manuals.htm>
- Tominaga, Y., Mochidab, A., Yoshiec, R., Kataokad, H., Nozue, T., Yoshikawaf, M., Shirasawa, T. (2008) AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. *J. Wind Eng. Ind. Aerodyn.* 96 1749-1761.