Simulating Longitudinal Ventilation Flows in Long Tunnels:
Comparison of full CFD and multi-scale modelling
approaches in FDS6

Chin Ding (Edmund) Ang\textsuperscript{a}, Guillermo Rein \textsuperscript{*b}, Joaquim Peiro\textsuperscript{a}, and Roger Harrison\textsuperscript{c}

\textsuperscript{a}Department of Aeronautics, Imperial College London
\textsuperscript{b}Department of Mechanical Engineering, Imperial College London
\textsuperscript{c}Fire & Risk, AECOM UK

November 2, 2015

Abstract

The accurate computational modelling of airflows in transport tunnels is needed for regulations compliance, pollution and fire safety studies but remains a challenge for long domains because the computational time increases dramatically. We simulate air flows using the open-source code FDS 6.1.1 developed by NIST, USA. This work contains two parts. First we validate FDS6’s capability for predicting the flow conditions in the tunnel by comparing the predictions against on-site measurements in the Dartford Tunnel, London, UK, which is 1200 m long and 8.5 m in diameter. The comparison includes the average velocity and the profile downstream of an active jet fan up to 120 m. Secondly, we study the performance of the multi-scale modelling approach by splitting the tunnel into CFD domain and a one-dimensional domain using the FDS HVAC (Heating, Ventilation and Air Conditioning) feature. The work shows the average velocity predicted by FDS6 using both the full CFD and multi-scale approaches is within the experimental uncertainty of the measurements. Although the results showed the prediction of the downstream velocity profile near the jet fan falls outside the on-site measurements, the predictions at 80 m and beyond are accurate. Our results also show multi-scale modelling in FDS6 is as accurate as full CFD but up to 2.2 times faster and that computational savings increase with the length of the tunnel. This work sets the foundation for the next step in complexity with fire dynamics introduced to the tunnel.

Keywords: Fire Dynamics Simulator, Computational Fluid Dynamics, Multi-scale, Tunnel Ventilation System.

1 Computer Modelling for Tunnel Ventilation

The Department for Transport (\cite{Department of Transport 1999}) classifies a covered roadway as a tunnel if it exceeds 150 m. However in practice, the majority of road and rail tunnels in the world far exceed 150 m, e.g. Mont Blanc tunnel at 11.61 km or the world’s longest Laerdal tunnel at 24.51 km. The significance of this classification is that tunnels require the provision of ventilation system for fumes and fire emergency.

Broadly, there are three mechanical options \cite{PIARC 1999} to ventilate a tunnel: a fully transverse, a semi-transverse and a longitudinal system. For a longitudinal system, air movement is

\*Please address all correspondence to Guillermo Rein currently at Imperial College London, United Kingdom. Email: g.rein@imperial.ac.uk Tel: +44 (0)20 7947 7029
induced in one direction along the tunnel via jet fans located along its length. This is unlike transverse systems where ducted extracts that take up valuable space are needed along the tunnel. Therefore, a longitudinal system is comparatively simpler to design and operate than the transverse systems.

Currently in the industry, there are two main computer modelling approaches for the design of a tunnel ventilation system: 1D and CFD approaches.

Firstly, the 1D approach for ventilation simulation in tunnels can be traced back to the late 70s (Greuer, 1985) where Greuer first developed a tool to simulate steady-state calculations in a tunnel system. A 1D model has a low computational requirement where it can be used to simulate a whole tunnel network including chimneys, main tunnels and branches. However, the key disadvantage is that it is useful for bulk flows, but does not adequately represent flows in complex regions or the couplings between multiple variables, e.g. jet fans, tunnel geometry, traffic, environmental influence, heat etc. With the increasing size and scale of the complexity of modern tunnels, the 1D method alone is increasingly inadequate to model these complex interactions and couplings.

On the other hand, CFD is a more modern technique where the first substantial work on CFD for tunnel ventilation can be traced back to Fletcher in the 90s (Fletcher and Kent, 1994), with additional work over the last two decades focused mainly on refining the numerical methods (Woodburn and Britter, 1996; Gao et al., 2004) and also on verification and validation of the solvers (McGrattan et al., 2014b). Although CFD allows better representation of complex regions, and for a larger variety of quantities to be coupled and modelled, this is not without a trade-off. As CFD is computationally expensive, for practicality CFD is currently used to only model a select region of a tunnel ventilation section. It does not allow the modelling of a longer tunnel section so the coupling or interaction, e.g. between tunnel section and a station, can be studied.

In an attempt to address this challenge, a third approach, i.e. the multi-scale method has been proposed. In general, multi-scale modelling has been considered on various fluid flow applications, e.g. to simulate blood flow (Formaggia et al., 2001). However, this method has largely been overlooked by the tunnel ventilation industry where prior to Colella et al (Collela et al., 2009), only Rey et al (Rey et al., 2009) have publicised the application of multi-scale fire modelling for tunnels.

Fundamentally, this hybrid approach uses CFD for complex regions and 1D for far-field regions where an area-averaged representation of the variables is acceptable. This approach is to allow the simulation of a larger model that is impractical using the full CFD method alone, while maintaining a practical computational cost and a reasonable accuracy in regions of interest to the designers.

As the work by Colella et al (Collela et al., 2009) on multi-scale modelling was based on a Reynolds-Averaged Navier-Stokes solver, the purpose of our work is to investigate the multi-scale modelling approach using FDS6, a Large Eddy Simulation based solver and to validate FDS6 against onsite experimental measurements.

2 Tunnel Ventilation Simulation Using FDS6

FDS (McGrattan et al., 2014) version 6.1.1 is the focus of this study where it is one of the well-known CFD solvers in the building industry. Although there have been various researches and experiments on FDS applications’ in buildings (McGrattan et al., 2014b; Lee and Ryou, 2006; McGrattan et al., 2014d), there have been comparatively limited outputs on FDS for modelling in tunnels, e.g. the FDS Validation Guide currently has only one validation case for a tunnel simulation study. In addition to the lack of validation cases, FDS is also not as widely adopted for tunnel applications compared to buildings applications.
Fundamentally, the mathematical model (McGrattan et al., 2014a) used in FDS is intended for low Mach (M < 0.3) flow, and therefore it can be applied for flows in a transport tunnel. From an industry perspective, the lack of adoption of FDS in tunnel applications is in large part due to the rectilinear grids requirement compared to other solvers where unstructured grids that better reflect an actual tunnel structure can be modelled.

To increase practitioners and researchers confidence on FDS’s capability for tunnel modelling applications, it is important to ensure FDS’s prediction reflect physical phenomena, i.e. it can reasonably predict the flow rates and flow profile in a real world tunnel. To do this, it is proposed in this study to validate and demonstrate FDS’s capability in predicting a cold flow (ambient) condition in the tunnel by modelling the predicted averaged velocities and velocity profiles in the tunnel and comparing to those measured on site in a real tunnel. This is considered appropriate as although the design process of a tunnel ventilation system involves various complex and intricate variables, fundamentally the most important aspect is to ensure the jet fans and flow rates perform as intended before additional complexities are introduced.

In addition to the above, the multi-scale modelling approach has recently been made possible on FDS using the HVAC feature where Izabella Vermesi of Imperial College London demonstrated the feasibility of using FDS6 for multi-scale modelling. The current work on multi-scale fire modelling in tunnels (Collela et al., 2011a, 2009, 2010b, 2011b) has been based on ANSYS FLUENT only. Considering the widespread adoption of FDS, it is proposed in this study to investigate whether FDS6 predicted average velocities using the multi-scale method are comparable to the full CFD models and the on-site measurement (validation).

The on-site data used here is obtained from Colella’s work (Collela et al., 2009) that is based on the UK Dartford Crossing (West tunnel) across river Thames that connects Kent (South) and Essex (North). The geometry of the tunnel is illustrated in Figure 1.

3 Multi-scale Method in FDS6

FDS is an open source CFD package developed by NIST and VTT, and was first introduced in February 2000 (McGrattan et al., 2014c). FDS has been developed to numerically solve thermally driven, low Mach number flow represented by a modified form of Navier-Stokes equations.

The core numerical scheme for FDS (McGrattan et al., 2014a, McDermott, 2014) is a second-order accurate explicit predictor-corrector scheme with a Cartesian grid, and turbulence is represented by the implicitly filtered LES with the turbulent viscosity represented by the Deardorff’s model.

![Figure 1: Dartford West tunnel key dimensions](Collela et al., 2009)
Details of the mathematical models are thoroughly documented in the FDS Technical Reference Guide (McGrattan et al., 2014a) and therefore have not been repeated here.

Multi-scale modelling using FDS is based on an indirect coupling method (Floyd, 2011) utilising the HVAC feature. Details of the HVAC implementation are in the FDS Technical Reference Guide (McGrattan et al., 2014a) which is based on the work by Floyd (Floyd, 2011) and only the key aspects are presented here.

The HVAC or 1D model (McGrattan et al., 2014a; Floyd, 2011) is represented as a network of ducts and nodes where the nodes represent the interface between the 1D and CFD regions. A duct represents a continuous flow path with various properties (areas, length, surface roughness) and is defined by two nodes, e.g. four nodes are needed to represent two ducts with different properties.

The conservation equations for the HVAC model are mass, energy and momentum (McGrattan et al., 2014a):

\[ \sum_j \rho_j u_j A_j = 0 \quad (1) \]
\[ \sum_j \rho_j u_j A_j h_j = 0 \quad (2) \]
\[ \rho_j L_j \frac{du_j}{dt} = (p_i - p_k) + (\rho g \Delta z)_j + \Delta p_j - \frac{1}{2} K_j \rho_j |u_j| u_j \quad (3) \]

where subscript \( i \) and \( k \) represent nodes, and \( j \) a duct segment. The right hand side of the momentum Equation (3) represents the pressure gradient between two nodes, buoyancy head, pressure rise due to fans and pressure losses due to wall friction, sharp turns and expansion or contraction. Given the tunnel’s length in the model, the wall friction constitutes most of the duct losses and is modelled as \( K = \frac{f L}{D} \), where \( L \) is the duct length and \( D \) is the diameter.

The friction coefficient, \( f \) is determined from the Colebrook equation which is approximated using the Zigrang and Sylvester method (McGrattan et al., 2014a) as:

\[ \frac{1}{\sqrt{f}} = -2 \log_{10} \left( \frac{\epsilon}{D} \frac{3.7}{Re_D} \log_{10} \left( \frac{6.9}{Re_D} + \left( \frac{\epsilon}{D} \frac{3.7}{3.7} \right)^{1.11} \right) \right) \quad (4) \]

where \( \epsilon \) is the absolute roughness. Currently, the HVAC duct does not allow the storage or delay of mass, i.e. the mass is conserved. The discretisation and solution procedure of the momentum equation are available in the FDS guide (McGrattan et al., 2014a) and are therefore not repeated here.

The HVAC model uses prior time step values as its boundary conditions and then provides FDS the wall boundary conditions including velocity, temperature and gas species for computation of the next time step in FDS (Floyd, 2011). The boundary conditions at the HVAC inlet node are determined by summing the mass and energy of the CFD cells next to the node and averaging the pressure (McGrattan et al., 2014a). These are then used to determine the average temperature:

\[ \bar{\rho}_i = \frac{\sum_j \rho_j A_j}{\sum_j A_j} ; \quad \bar{Y}_{\alpha,i} = \frac{\sum_j Y_{\alpha,j} \rho_j A_j}{\sum_j \rho_j A_j} ; \quad \bar{P}_i = \frac{\sum_j P_j A_j}{\sum_j A_j} \quad (5) \]
\[ \bar{h}_i = \frac{\sum_j \rho_j A_j c_p(T_j, Y_j)}{\sum_j \rho_j A_j} ; \quad \bar{T}_i = \frac{\bar{h}_i}{c_p(T_i, Y_i)} \quad (6) \]
where $i$ is the node and $j$ is the CFD cells next to the node. $Y$ is gas species mixture fraction, $h$ is enthalpy and $W$ is molecular weight.

At the HVAC node outflow, the CFD wall cells boundary conditions are based on the flows obtained from the HVAC model. The surface temperature, $T_w$ is based on the value in the connected HVAC duct. Mass flow, the other wall boundary condition is computed as (McGrattan et al., 2014a):

$$\dot{m}_w'' = Y_{\alpha,d} \dot{m}''_\alpha$$
$$\dot{m}''_\alpha = u_d \rho_d A_d$$  \hspace{1cm} (7)

where $A_v$ is the total vent (node outlet) area. Subscript $d$ represents the duct and $w$ represents the wall cells. With the two information (McGrattan et al., 2014a), the wall cells boundary conditions (velocity, density and gas species) are obtained iteratively with 20 maximum iterations based on:

$$u_w = \frac{\dot{m}''_\alpha}{\rho_w}; \quad \rho_w = \frac{p W}{RT_w}$$  \hspace{1cm} (8)

$$Y_{\alpha,w} = \frac{\dot{m}''_\alpha + 2 \rho_w \frac{DY_{\alpha,gas}}{\delta n}}{2 \rho_w \frac{D}{\delta n} + u_w \rho_w}$$  \hspace{1cm} (9)

The coupling between 1D and CFD is relatively simplistic and does not include any measure of turbulence intensity. Therefore, the 1D interface should be sufficiently away from regions with turbulent flows. Separately, flow through the HVAC model is forced in uni-direction, with flow reversal flow possible only at the interface.

4 Dartford West Tunnel Model

For the onsite data on Dartford West Tunnel (Collela et al., 2009), measurements were obtained for the bulk flow generated by different combination of jet fans pairs and also on the velocity at different height at various distances from an individual jet fan. The tunnel is 1.5 km long and with an effective area of 41 m$^2$. Figure 2 shows the key dimensions of the tunnel.

There are 14 jet fans pairs (Collela et al., 2010b) in total where 7 is located at the South portal (nearest fan is 100 m) and 7 is located at the North portal (nearest fan is 50 m). These fans are spaced 50 m apart in series, individual fan is spaced at 1.2 m in parallel and are mounted at 5.5 m above road. Each fan has a diameter of 0.5 m$^2$ and generates a volume flow rate of 8.9 m$^3$/s.

There are two chimneys located before the jet fans at South portal and after the jet fans at North portal. These have been omitted in FDS as these are unlikely to significantly impact on the modelling given their positions relative to the jet fans, and also due to the lack of detail information on these chimneys.

As FDS is based on a rectilinear grid, the tunnel is represented as a straight tunnel of a square 6.4 m $\times$ 6.4 m (40.96 m$^2$) with no incline in order to reduce the complexity for this study; this is in line with work by Collela et al. (Collela et al., 2010b).

The fact that the tunnel is square, instead of round is due to the inherent rectilinear grid restriction of FDS. For a low Mach flow, this approximation is applicable for studying the global effects and bulk flows in a tunnel, which is the main objective of this study. Also, if the tunnel is constructed, using rectilinear blocks to mimic the curved structure, the construction blocks would form a stair-stepped sawtooth and this creates an unrealistic flow and drag near the wall boundary. To demonstrate the approximation is reasonable, a CFD study comparing the square duct tunnel and a tunnel with stair-stepped geometry to replicate the curvature of the actual tunnel has bee
Figure 2: Dartford West tunnel cross section (Collela et al., 2009)

Figure 3: On left is the front view of the stair-stepped tunnel. On right is the comparison of the mass flow out of the square and stair-stepped tunnels.
carried out. As shown in Figure 3 below, the results show similar mass flow (approximately 2% difference) in the tunnel despite the different geometry.

The tunnel walls are modelled as concrete with an absolute roughness, $\epsilon = 0.02$ m derived from a typical tunnel friction coefficient, $f = 0.026$ (Jang and Chen, 2002) as the actual tunnel’s properties are unknown. The same $\epsilon$ is specified for the 1D region.

Beyond the portals, the mesh is extended to 10 m on each side to simulate the atmospheric environment instead of terminating the mesh directly at the portal.

As the experimental measurements were taken when the tunnel was closed, i.e. no traffic, this obstruction free scenario is also reflected in the model.

5 Jet Fans and the Background Velocity

Prior to the introduction of the HVAC feature, jet fans in FDS are modelled by specifying a velocity value on a vent at the same height as a jet fan. This is fundamentally incorrect as the blowing vent would inject additional mass into the compartment in order to generate a velocity whereas a real jet fan is designed to induce flow by increasing the pressure of the air flowing through it, i.e. it does not inject additional mass into the flow.

The HVAC feature can be used to model a jet fan. Although it does not allow a pressure to be specified for a flow across, a volumetric flow rate can be specified across two connected nodes thereby simulating a jet fan’s intake and outlet. 8.9 $m^3/s$ is specified for each fan as the West tunnel specification and the fan start up is based on the FDS default instantaneous ramp-up time. Although the FDS User Guide (McGrattan et al., 2014c) provided a coded example, practitioners will need to model an enclosure around the HVAC nodes.

The fan shroud is modelled as an enclosure around the HVAC nodes, with the volumetric flow rate applied at the middle section. Because of the 0.4 m grid, the area of the modelled jet fan is 0.32 $m^2$, 0.4 m (h)/0.8 m (w)/1.6 m(l), where the actual jet fan is 0.5 $m^2$. Note that as the length is based on a generic jet fan, coupled with the approximation between a circular and square tunnel, when the velocity profile downstream of the modelled jet fans is compared to those by Colella (Collela et al., 2010a), it is expected that there will be discrepancies particularly at turbulent regions near the jet fans’ discharge.

Average background flow of 1.7 m/s as per the on-site data (Collela et al., 2010b) is achieved with two 10 m/s slot fans, 0.8 m (h)/6.4 m (w)/0.4 m (l), located at 0.4 m from the ceiling and floor. The fans are positioned at an arbitrary 20 m from the South portal, and not directly by the portal to minimise sudden contraction losses.

Two slots are used to shorten the downstream it takes to generate a fully developed flow profile to ensure the flow is fully developed before the first jet fan pairs. The 10 m/s needed to achieve the 1.7 m/s average background flow is from trial and error.

6 Multi-scale Model and Interface Locations

A key consideration for a multi-scale model is the length, $L_{3D}$ of the CFD domain (near field), i.e. regions where CFD is needed to resolve the flow’s characteristic. Once this is decided, the remaining areas are treated as far field regions where these can be represented in 1D.

For a cold flow only scenario, the CFD domain is the region where the jet fans are located, with the far field domain modelled in 1D. Although not part of this paper, when a fire is introduced, the fire will also be located in a CFD domain given its complex reaction. By modelling the jet
fans and the fires in CFD regions together with the regions in between represented with 1D, this essentially couples together the jet fans and the fires, allowing the effect of this coupling to be simulated.

To establish the criteria to determine the length of the CFD and 1D domains, the following supplementary criteria appropriate for FDS is needed (read in conjunction with Figure 5):

1. **Length upstream,** $L_{JF,UP}$ and **downstream,** $L_{JF,DW}$ of jet fans. To determine these two lengths, a separate model with only the background velocity and 1 jet fan pairs has been modelled. The model is intended to assess the distance needed upstream (intake) and downstream (outlet) from the jet fans for the flow to be fully developed.

   See Figure 4. A downstream distance, $L_{JF,UP} = 130$ m is needed for the flow to be fully developed. On the upstream, the flow becomes fully developed at 35 m (when no jet fans are located within 130 m upstream). Therefore, $L_{JF,UP} = 35$ m. These distances are specific to this study only, and will need to be reassessed for others.

2. **Length from portal,** $L_P$. An arbitrary 50 m is maintained between a portal and the 1D node as an error was returned when the 1D interface is located at the mesh boundary.

Using the parameters above, the CFD regions in the multi-scale model can be configured as shown in Figure 5. The length of the CFD region, is summarised in Table 1. Note that as the 1D duct is between the CFD regions, the total length of the CFD region, $L_{3D} = L_1 + L_2$.

With more jet fans pairs, the length of the 1D region decreases. However, this is only an issue for a shorter tunnel where in a longer tunnel, comparatively greater length in the tunnel can be represented by 1D regions after factoring in the 35 m and 130 m interface locations.

The onsite measurement and the model considered are obstruction free. However in practice, modelling of tunnel ventilation will often need to account for the presence of vehicles (obstruction).

### Table 1: Length of 1D and CFD regions in multi-scale models. Note that BV is background velocity and JFP is jet fans pairs.

<table>
<thead>
<tr>
<th>Scenarios</th>
<th>Length of CFD region, $L_{3D}$ (m)</th>
<th>Length of 1D region (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>BV only</td>
<td>220</td>
<td>1,280</td>
</tr>
<tr>
<td>1 JFP + BV</td>
<td>280</td>
<td>1,220</td>
</tr>
<tr>
<td>4 JFP + BV</td>
<td>430</td>
<td>1,070</td>
</tr>
<tr>
<td>7 JFP + BV</td>
<td>580</td>
<td>920</td>
</tr>
<tr>
<td>14 JFP + BV</td>
<td>930</td>
<td>570</td>
</tr>
</tbody>
</table>
When the multi-scale method is considered, obstructions may be modelled in the 1D region by reducing the diameter to simulate the presence of an obstruction or by specifying the loss in the 1D region. For obstructions in the CFD region, it is suggested to maintain an arbitrary 50 m minimum distance between the obstruction and the 1D and CFD interface. Future researches may be able to better inform the optimum approach when including obstructions in multi-scale modelling.

7 Cold Flow Simulation Results

Full CFD and multi-scale models for the Dartford West tunnel cold flow (no fire) ventilation have been carried out, and compared with the on-site measurement obtained by Collela et al (Collela et al., 2010b). Five scenarios have been modelled where these are: background velocity only, one, four, seven and fourteen jet fans pairs activated. The average velocity at the centre point of the tunnel in these five scenarios are then compared to the on-site measurement.

Figure 6 shows the comparison of the on-site and both 1D and full CFD modelling average velocities at the centre section of the tunnel.

Referring to left of Figure 6, the full CFD and multi-scale models show good agreement and this
suggests the multi-scale models are capable of achieving similar results to those of the full CFD models. However, the modelled velocities are at the lower end of the measurements. The more probable reason for this is that the assumed surface friction factor does not reflect the actual surface friction factor of the tunnel.

This is shown in right of Figure 6 where two additional models (4 and 14 jet fans pairs) have been simulated but with negligible surface friction in the tunnel. This shows the surface roughness in the tunnel is most likely between 0.001 m and 0.02 m, and also highlights the importance of using a relevant surface friction value in a model.

Additional simulations have also been carried out to compare the velocity profile generated by a jet fan along the tunnel at various distances with those measured on-site shown in Figure 7. The on-site measurement and the FDS model are based on a jet fans pair and background velocity, and the distances considered as 20 m, 40 m, 60 m, 80 m, 100 m and 120 m downstream of the fan. The locations of the measurements are taken along two profiles (see Figure 2): Profile 1 (centre of tunnel) and Profile 2 (2.5 m from the centre of tunnel).

Figure 7 shows the velocities at 20 to 60 m have weak correlations to the on-site data below 4.5 m. As the flow is fully developed further downstream of the jet fans (this can be observed by the smaller change in the velocity gradient along y-axis), the velocity profiles correlate with the on-site measurement.

This weak correlation is due to the complexities on how the jet fan is modelled. The intention of this study is not to calibrate the flow characteristic near the jet fans, but to validate the bulk flow of these jet fans. Therefore, this weak correlation does not adversely affect the objective of this study.

To demonstrate the complexity and sensitivity of the velocities profiles to changes in how the jet fans are modelled, two different jet fans have been modelled. On the left of Figure 8, the fan shroud area has been doubled to 0.64 m$^2$ and this shows an increase in the velocity at 4.5 m, but do not significantly change the profile below 4.5 m.

On the right of Figure 8, the flow rate is increased to 10 m$^3$/s and the fan shroud has been removed so the flow disperses instead of forming a jet flow. Although the velocity profile better correlates with the measurement, this requires the volume flow rate of the fans to be increased beyond the Dartford tunnel’s specification of 8.9 m$^3$/s.

The two additional models are intended to show the complexity of modelling the jet fans, and although it is possible to calibrate the fans to match the on-site measurement, this requires modifying the jet fan’s to beyond those provided in Dartford tunnel.

8 Computational Time

In addition to the validation study, the other objective is to assess the benefit multi-scale method to reduce the computational time to enable modelling of larger tunnel networks. Timings from the multi-scale model show the HVAC section uses less than 1% of the overall CPU time. Therefore, as the computational cost for HVAC (1D) is negligible, any reduction in CFD regions will reduce the run time.

Based on this paper’s work, Figure 9 shows a reduction in computational time up to 2.2 times depending on the extent of the CFD regions. It is envisaged the multi-scale method will provide greater improvement in computational time in longer tunnel as the ratio of 1D to CFD region will be greater.
Figure 7: Comparison of the velocity profiles downstream from 1 jet fan pair (background velocity included) with on-site measurements. The velocities are taken at steady-state and are based on a 20 seconds average. Profile 1 (centre of tunnel) and Profile 2 (2.5 m from the centre of tunnel).
Figure 8: Comparison of the velocity profiles downstream from 1 jet fan pair (background velocity included) with on-site measurements. The left figure shows the profile when the fan area is doubled to 0.64 m²; the right figure is when the fan shrouds are removed and the volume flow rate is increased by 10% to 10 m³/s. The velocities are taken at steady-state and are based on a 10 seconds average. Profile 1 (centre of tunnel) and Profile 2 (2.5 m from the centre of tunnel).

Figure 9: Computational performance between full CFD and multi-scale models. The figure shows the comparison of computational hours between full CFD and multi-scale to 250 seconds.

9 Conclusions

Full CFD and multi-scale FDS simulations have been compared against on-site measurements for Dartford West tunnel. Although the predicted average velocities for both methods are at the lower end, all results correlate with the measurement.

The jet fan’s predicted velocity profiles have also been compared to the on-site measurement between 20-120 m (20 m interval) downstream. Although the predicted velocity profiles show
good correlation at 80 m and beyond downstream, they fall outside the measured values at 60 m and less, in particular for the values measured at less than 4.5 m above floor level.

This is attributed to the complexities of modelling the jet fans when the details of their properties are not known, and not a weakness in the FDS solver as any changes introduced, e.g. changing fan shroud size or volume flow rate, can drastically change the velocity profiles. Further calibration of the model could be considered to improve the correlation of the modelled jet fans to the on-site measurement.

The multi-scale method has shown a significant reduction in computational time of up to 2.2 times compared to full CFD. The minimum length of the CFD region in a multi-scale model based on Collela et al. [2010b] has also been modified to include additional lengths of the CFD regions upstream and downstream of the jet fans, and the length of CFD region from the portal. These lengths are needed when applying the multi-scale approach using FDS. Further work can be considered to reduce the minimum upstream length of the CFD region for the jet fans, particularly the length of the CFD region from the portal as the 50 m is arbitrary and there is potential to eliminate this.

The study in this paper shows FDS as a solver is capable in predicting to reasonable accuracy the bulk flow velocity in a tunnel using both the CFD and the novel multi-scale approaches. With FDS’s predictive capability in a tunnel demonstrated, this work sets the foundation for the next step in complexity when fire dynamics, which is the key application of FDS, are introduced to the tunnel.

References


