INVESTIGATION OF A COMPUTATIONALLY EFFICIENT MULTI-SCALE FIRE MODELLING METHOD IN LONGITUNDIALLY VENTILATED TUNNELS FOR FDS6.1

Chin Ding (Edmund) Ang, Guillermo Rein, Joaquim Peiro, Roger Harrison, and Izabella Vermesi

[1] Imperial College London
chin.ang13@imperial.ac.uk
[2] Imperial College London
g.rein@imperial.ac.uk
[3] Imperial College London
j.peiro@imperial.ac.uk
[4] AECOM UK
roger.harrison@aecom.com
[5] Imperial College London
i.vermesi14@imperial.ac.uk

Abstract. To reduce the computational time for modelling large tunnel networks, we used FDS6.1 to investigate the solver's capabilities for multi-scale modelling by combining 1D and full CFD models in the same domain. This study is to investigate the capabilities of FDS6.1 for simulating a tunnel ventilation system (cold and fire) using the multi-scale method. Firstly, we replicated the physical condition for cold flow using the multi-scale method and we validated the predictions against field measurements in the Dartford Tunnel, UK. This study demonstrated both the multi-scale and full models have good correlation with each other, and with the field measurements. Depending on the ratio of 1D and full CFD plus the scenarios modelled, the reduction in the computational time for the multi-scale method varies from 20% to 220% compared to the full CFD method. Secondly, we introduced a fire up to 75 MW in the multi-scale and full CFD models. Both models exhibit oscillatory mass flows along the tunnel similar to the pulsation effect observed in a tunnel fire. In the multi-scale method, the models significantly under or over-predicted the mass flows.

1. RESEARCH INTRODUCTION

With exponential increase in population growth and development of ever larger densely populated cities, tunnels are often the only viable option to expand the transport infrastructure connecting towns and cities. Construction of modern road and rail tunnels increasingly becomes longer, larger and intricately connected. For example, Laerdal Tunnel is currently the world's longest tunnel at 24.51 km.

With an increasing number of long tunnels used for transport infrastructure, these tunnels need to be adequately ventilated to ensure tunnel users will not be negatively impacted by fumes from vehicles or by smoke in a fire. Although there are various options to provide ventilation in a tunnel, these methods can be broadly categorises into longitudinal, transverse or semi-transverse ventilation. In design and engineering, the ventilation system in tunnels are currently designed using both 1D and full CFD modelling methods.

1D is used at the early stages of a project where time and resources are constrained, whereas full CFD modelling is used at the later stages to validate the designs informed by the 1D modelling method. Although full CFD modelling provides detailed results, this method is significantly constrained by the computational time which can take from days to weeks. In addition, to ensure the computational time is kept to a realistic time frame for the design and engineering process, only a section instead of the larger tunnel network is modelled. This shows the current limitations for the practical application of the full CFD method.

With the multi-scale method, this is a hybrid approach combining CFD for complex regions and 1D for far-field regions of a tunnel section where an area-average representation of the variables is acceptable. This approach allows designers to simulate a significantly longer tunnel section that is impractical using only either the full CFD, or the 1D method due to the constraints of computational time.

In this research, we have used FDS6.1 [3] and the built-in HVAC feature to implement the multi-scale modelling method. The tunnel model in this research has been based on the Dartford West Tunnel and the onsite cold flow ventilation data (velocity) collected by Collela *et al* [2]. The onsite cold flow data is used to determine whether the multi-scale modelling method has a good correlation with both the field data and the predictions from the full CFD method. This is important to demonstrate the predictions obtained from the multi-scale modelling method correlates with real world observation. The computational times from modelling the cold flow ventilation of the Dartford West Tunnel have also been used to assess the reduction in computational time between the full CFD method and the multi-scale modelling method.

Following on that, we have used the same tunnel models but this time included a fire up to 75 MW in both the full CFD and multi-scale models to consider the feasibility of modelling a tunnel fire using the multi-scale modelling method.

For the full CFD models, a grid sensitivity study and a non-dimensionalised validation study have also been considered. We have benchmarked the full CFD models against the ArupFire Tunnel Fire Experiments documented in the FDS Validation Guide [4]. The results of the validation study has not been presented here.

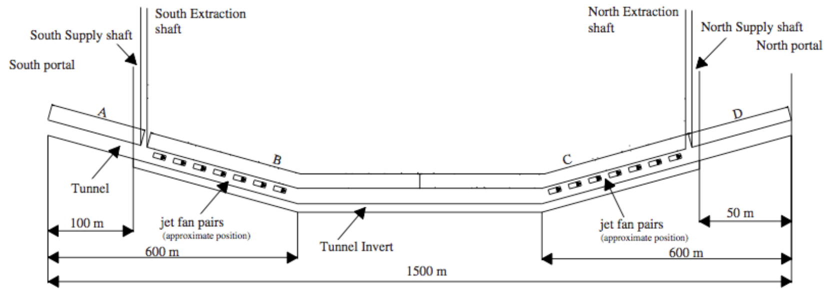

Figure 1. [Dartford West Tunnel Long Section. 2]

Note that we have previously published the cold flow validation study in a separate peer reviewed journal [1]. The results from the multi-scale fire modelling are first published here.

2. THE DARTFORD TUNNEL, UK

The Dartford Tunnel is 1.5 km long with an effective area of 41 m^2. There are 14 jet fans pairs in total where 7 is located at the South portal (100 m) and 7 is located at the North portal (50 m). These jet fans pairs are spaced 50 m apart in series, and the individual fans is spaced at 1.2 m in parallel and are mounted at 5.5 m above the road. Each fan has a diameter of 0.5 m^2 and generates a volume of rate of 8.9 m^3/s. See Figure 1.

3. MULTI-SCALE MODELLING METHOD

3.1. SUMMARY OF THE MULTI-SCALE MODELLING METHOD

We have used FDS6.1 (current at the time the work was carried out) and the built in HVAC feature to implement the multi-scale modelling method. Briefly, multi-scale modelling can be divided into two methods, namely the direct and indirect method [1].

A key consideration for a multi-scale model is the length 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. 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.

See Figure 2 for the implementation of the multi-scale model. Note that L_P is 50 m, $L_{JF,DW}$ is 130 m and $L_{JF,UP}$ is 35 m. We have derived these distances by calibrating the CFD models.

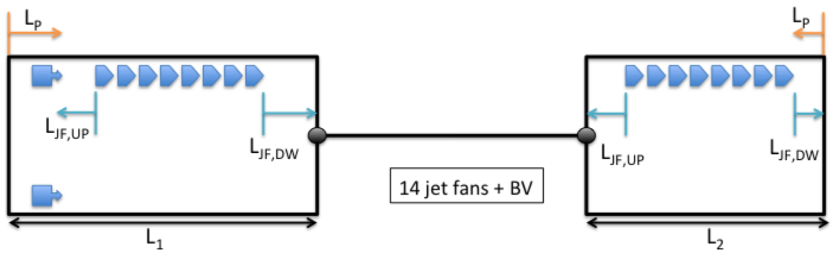

Figure 2. [Illustration of the Implementation of the FDS6.1 Multi-scale Model. 1]

3.2. COMPARISON OF NUMERICAL MODELLING TO FIELD DATA

To compare the numerical model for cold flow multi-scale modelling against field data, we have carried out modelling for five scenarios including one with only background velocity (no jet fans active), and models with one, four, seven and fourteen jet fans pairs activated [1].

Figure 3 shows the comparison of the on-site and both 1D and full CFD modelling average velocities at the centre of the tunnel. Although the velocities at 20 m to 60 m have weak correlations to the field data, the velocity profiles further downstream of the jet fans correlate with the field data. This is within expectation as the numerical modelling of the jet fans are highly sensitive, particularly when considering the velocities profile near to the jet fans (highly turbulent and sensitive to the configuration of the jet fans). As the results show, as the flow from the jet fans becomes fully developed, the velocity profiles are within the field measurements.

The results show for bulk flow in a tunnel, the multi-scale modelling method provides reasonably accurate predictions when compared to field measurements. Separately, the multi-scale modelling method also provides significant time saving compared to the full CFD method.

Figure 4 shows the computational time between the full CFD method and the multi-scale modelling method where depending on the length of the 1D zone, where the larger a proportion of the model is represented in 1D against 3D, the more significant is the time savings, i.e. up to 2.2 times [1].

3.3. FIRE MODELLING

Following the demonstration of the accuracy of the numerical predictions of the cold flow multi-scale model to field measurements, we have carried out additional multi-scale model by introducing a fire in the centre of the tunnel. The model includes three fire sizes, i.e. 35 MW, 55 MW and 75 MW.

Implementation of the multi-scale model is similar to that of the cold flow modelling with the exception the fire is located in the 3D section. See Figure 5. Note that L_{fire} is 170 m and this has been determined based on the calibration

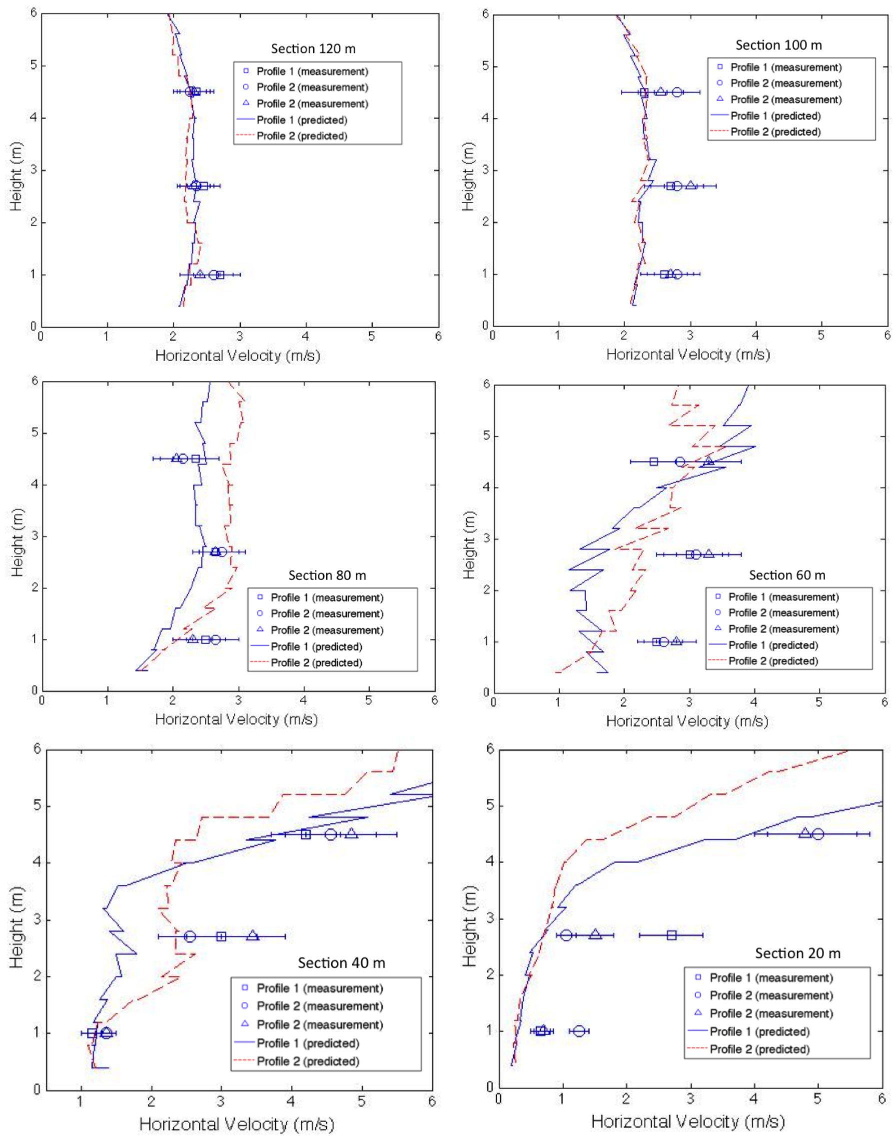

Figure 3. [Comparison of the Velocity Profiles Downstream of the Jet Fans. 1]

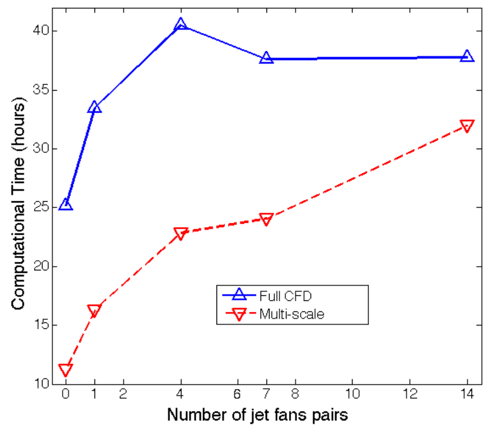

Figure 4. [Computational Time between Full CFD and the Multi-scale Modelling Method. 1]

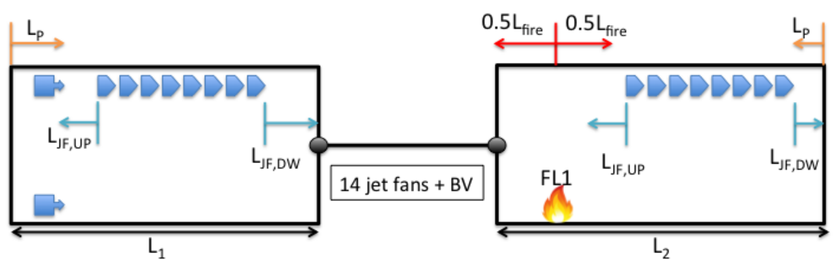

Figure 5. Implementation of the Multi-scale Model

of the model to ensure the fire has enough length to be fully developed when the smoke plume reaches the 1D section.

We have plotted the results from the full CFD and multi-scale models by considering the mass flow rates along the length of the tunnel at a specific time frame (50 and 250 seconds) and the mass flow rates in and out of the tunnel. As shown in Figure 6, the results show the mass flow along the tunnel for the multi-scale model is either under or over-predicted. Furthermore, the results show for the full CFD models, the mass flow reached a steady state whereas the mass flow for the multi-scale model continues to either increase or decrease overtime.

Since the work carried out in this research, we understand there has been development and improvement made in the implementation of the HVAC models. It is worth re-assessing the issue of the inconsistent mass flow rates in the tunnel. Because of this yet to be explained phenomenon, we have not assessed the computational time of the multi-scale and full CFD methods.

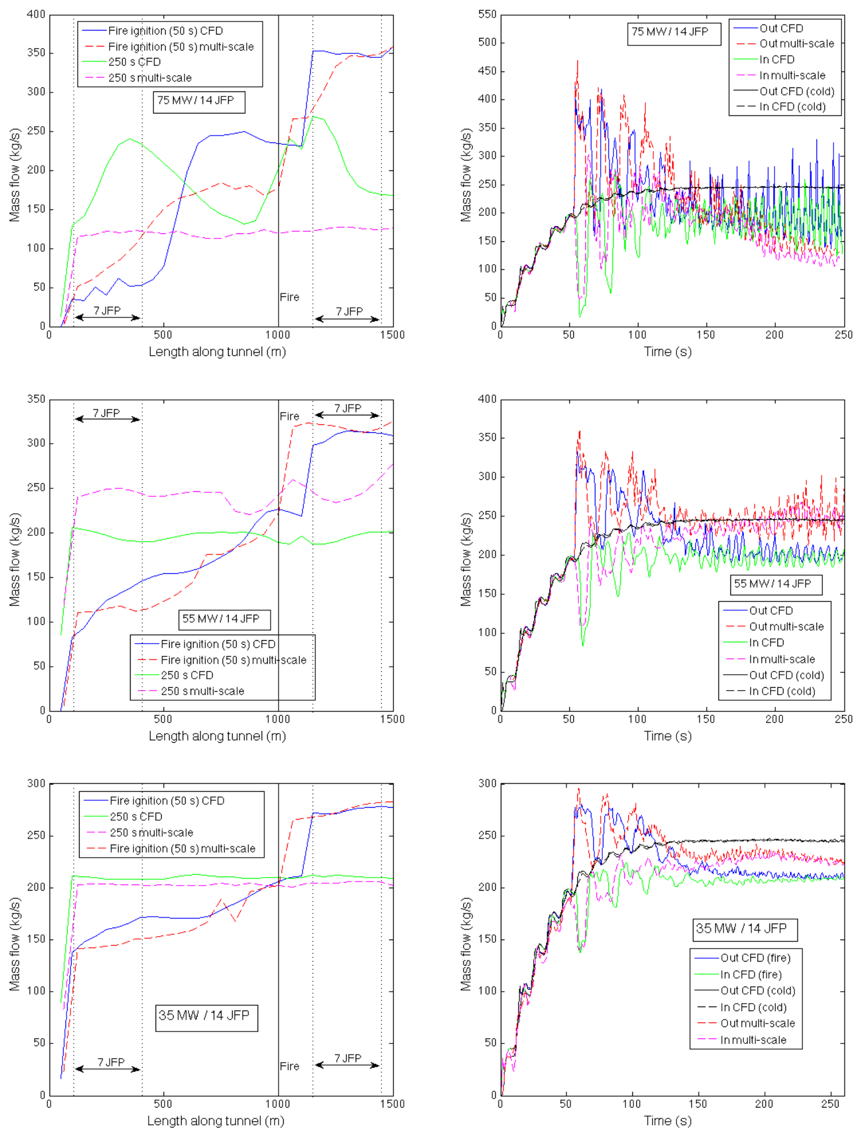

Figure 6. Mass Flow Rates for 35 MW, 55 MW and 75 MW fire.

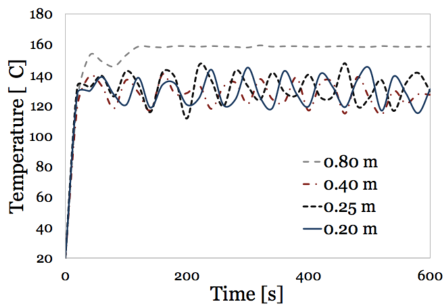

Figure 7. [Oscillating Temperature and Velocity. 5]

Separately, we observed as per Figure 6 the mass flow rates over time oscillate with a consistent wavelength and frequency. A similar observation has also been made separately by Vermesi *et al* [5]. The question on whether this fluctuation is a numerical or physical phenomenon is currently unanswered and requires further work. See Figure 7.

4. CONCLUSION AND RECOMMENDATIONS

We have shown in this work for cold flow modelling, it is possible to implement a multi-scale model in FDS6.1. Furthermore, the predictions obtained for cold flow modelling using the multi-scale modelling method are comparable to the field measurements obtained. In addition, depending on the ratio of the 1D against 3D sections, the savings in computational time for cold flow modelling implemented using the multi-scale modelling method can be up to 2.2. times.

Separately, we have shown for fire modelling using the multi-scale modelling method, there exists a yet to be explained phenomenon whereby the mass flow rates in the tunnel are under or over predicted when compared to the full CFD method. For further work, we recommend re-assessing this using the more recent build of the FDS where the HVAC model has been improved.

At this juncture, until the phenomenon of the mass flow rate predictions can be adequate explained, we advised against using the multi-scale modelling methods for fire modelling. For cold flow modelling, the work here shows the multi-scale modelling methods can be applied.

REFERENCES

[1] C.D.E. Ang, G. Rein, J. Peiro, and R. Harrison. “Simulating Longitudinal Ventilation Flows in Long Tunnels: Comparison of Full CFD and Multi-Scale Modelling Approaches in FDS6.” _*Tunnelling and Underground Space Technology*_ 52 (February): 119–126. Feb. 2016.

[2] F. Collela, G. Rein, and J. L. Torero. “A Novel Multiscale Methodology for Simulating Tunnel Ventilation Flows During Fires.” _*Fire Technology*_ 47 (January): 221–253. Jan. 2011.

[3] K. McGrattan, S. Hostikka, R. McDermott, J. Floyd, C. Weinschenk, and K. Overholt. _*Fire Dynamics Simulator User’s Guide*_. Sixth. NIST and VTT Technical Research Centre of Finland. May 2014.

[4] K. McGrattan, S. Hostikka, R. McDermott, J. Floyd, C. Weinschenk, and K. Overholt. _*Fire Dynamics Simulator Technical Reference Guide Volume 3: Validation*_. Sixth. NIST and VTT Technical Research Centre of Finland. May 2014.

[5] I. Vermesi, G. Rein, F. Colella, M. Valkvist, and G. Jomaas. “Reducing the Computational Requirements for Simulating Tunnel Fires by Combining Multiscale Modelling and Multiple Processor Calculation.” _*Tunnelling and Underground Space Technology*_. 2016.