Capabilities and Limitations of the Fire Dynamics Simulator in the Simulation of Tunnel Fires with a Multiscale Approach

Matteo PACHERA¹, Xavier DECKERS ¹² and Tarek BEJI²

¹ Fire Engineered Solution Ghent, Ghent, Belgium
² Ghent University-UGent, Dept. Flow, Heat and Combustion Mechanics, Belgium

mp@fesg.be

ABSTRACT

In tunnel fire safety engineering, long tunnels are generally simulated with one-dimensional fluid dynamics models (1D). Short tunnels can be simulated with a three-dimensional fluid dynamics model (3D). The 1D simulations are faster than 3D simulations, but they lack accuracy in the region where the flow is three dimensional, which is typically around the fire source. In this paper, we present a multiscale approach where large parts of the tunnel are simulated with the 1D approach, while a limited portion, close to the fire, is simulated with a 3D model. This allows to simulate with a good accuracy smoke dynamics and the induced flow field near the fire whilst large parts of the ventilation system and the tunnel’s structure are simulated with 1D, reducing thus the computational costs.

The present paper explores the capabilities and limitations of the Fire Dynamics Simulator, FDS 6.6.0, in the simulation of tunnels with a multiscale approach. FDS has an embedded 1D model connected with the hydrodynamics solver (3D) which allows to implement the multiscale approach without making a link to external models.

In the first part of the paper, the fan-induced flow field is simulated with the 3D model in order to generate an operative map where the pressure rise induced by the fan is a function of its volume flow rate. The operative map is later implemented in the 1D model without the need to model explicitly jet fans in FDS.

The second part of the paper addresses the modelling of a fire in a tunnel. The fire has two effects on the response of the ventilation system: (1) local pressure losses near the fire section and, (2) higher pressure losses along the tunnel due to the high temperature of the combustion products. Very promising results were obtained with the multiscale approach. Nevertheless, the results show also that, in the 3D portion of the tunnel (where the fire is located), large velocity fluctuations may lead to unphysical pressure distributions, especially for large fires (above 3 MW) and when a fine mesh is used. Therefore, further analysis and investigation is required in order to be able to deal with higher fire loads such as a 100 MW HGV fire or a 200 MW tanker fire.
INTRODUCTION

A multiscale approach allows to couple models, with different accuracies and complexities, in order to reduce the computational cost and maintain a high quality of the calculation. In the field of fluid dynamics, the multiscale approach has been applied to complex networks where one-dimensional (1D) and three-dimensional (3D) models are linked together. For tunnel problems, different authors implemented the multiscale approach to study the flow field in cold and warm conditions [1-4]. Large parts of the tunnel are simulated with a 1D model due to its lower computational cost compared to a 3D model. However, 1D models may not be suitable to describe complex flows that are actually 2D or 3D. Therefore, the region of interest should be simulated with a different model. In case of a tunnel fire, the flow field near a fire is fully 3D as well as the flow near a ventilation device (e.g., Saccardo nozzle, Jet fan and supply or exhaust vents). When smoke movement in the tunnel is the main focus of the study, ventilation devices can be included in the 1D network, by including their operative map. This can be calculated by resorting to Computational Fluid Dynamics (CFD) simulations or by estimating an installation efficiency [5,6].

MULTISCALE MODELLING IN FDS

For the simulation of tunnels with a multiscale approach the Fire Dynamics Simulator (FDS 6.6) is chosen because it already contains a Heat Ventilation and Air Conditioning (HVAC) model that can be used to simulate the 1D portions of the tunnel. The HVAC model in FDS has some limitations: (1) there is no thermal model that takes into account the temperature losses along the tunnel, and (2) the pipes have a constant section. However, the direct coupling of the hydrodynamic solver with the HVAC solver allows to update the solution at every time step. The boundary conditions are calculated at the interface between the 1D and the 3D domains. The quantities calculated by the 1D model are imposed uniformly at the interface of the 3D domain. The quantities calculated in the 3D domain are averaged over the surface before being transferred to the HVAC model [7]. At the boundaries of the 3D domain, the mass flow and the velocity are imposed as boundary conditions, while at the boundaries of the 1D domain the pressure is imposed.

Since the boundary conditions calculated by the 1D model are set uniformly at the interface of the 3D domain, the location of such interface is a key factor for the correct implementation of the multiscale approach. If the flow field in the 3D domain is not uniform over the section of the tunnel, the boundary conditions might affect the results of the simulation.

The tunnel under investigation has a square section of 25 m² and an overall length of 600 m. The tunnel is equipped with two jet fans with an area of 1 m² and a discharge velocity of 30 m/s, located 100 m from the inlet portal. The walls have a roughness of 0.003 m uniformly along the tunnel, the roughness is calculated considering a friction factor of 0.02, typical for road tunnels. The fire is located at 300 m from the inlet portal, in the middle of the tunnel. This location allows to freely move the interface between 1D and 3D domains without crossing the portals.

SIMULATIONS IN COLD FLOW

For the ventilation of the tunnel, two jet fans are installed in order to induce the longitudinal flow. The jet fans are first simulated with a 3D simulation and later, their operative map (i.e., pressure rise as function of volume flow rate) is implemented in the 1D model. The pressure rise induced by a jet fan in a tunnel is usually calculated according to Eq. 1 [5,6].

\[ \Delta P = \rho \cdot n_F \cdot k \cdot \frac{A_F}{A_T} \left( u_T - u_F \right) u_F \]  

(1)

Where \( n_F \) is the number of jet fans, \( k \) is the installation efficiency, \( A_F \) is the area of the fan, \( A_T \) is the area of the tunnel and \( u_T \) and \( u_F \) are the discharge velocities of the fan and the mean velocity in the tunnel. It is important to note that Eq. 1 represents the performance map of the jet fans in the tunnel, which is different from the performance map of an ‘isolated’ jet fan (as shown in Figure 4 for example). In this study, the jet fan is assumed to operate with a constant discharge velocity.
In the numerical model, the jet fans have a square section and are directly attached to the ceiling. This simplification is required in order to avoid the difference between models when using different mesh resolutions.

**Effect of the grid resolution**

Before calculating the operative map of the jet fan in the tunnel section, a preliminary mesh sensitivity analysis is performed. The results from three different mesh resolutions (see Table 1) are compared in order to find a suitable mesh size. The jet fans are simulated in a 80 m-long portion of the tunnel, where the total pressure is imposed at the inlet equal 0 Pa and a velocity of 3 m/s is imposed at the outlet. The latter value is chosen based on an estimation of the critical velocity obtained with the equation of Li [8].

![Pressure along the tunnel for different grid resolutions](image1.png)

![Pressure along the tunnel for different domain’s lengths](image2.png)

Figure 2: Pressure along the tunnel for different grid resolutions.

Figure 3: Pressure along the tunnel for different domain’s lengths.

The pressure rise between the inlet and the outlet of the tunnel, obtained with different grid’s resolutions, is similar, with a difference lower than 2 Pa (see Table 1). However, the pressure distribution predicted by Mesh 1 is different than the one obtained with Mesh 2 and 3 (see Figure 2). Therefore, Mesh 2 is chosen for the next simulations. The local pressure drop occurring at the fan’s section is consequence of the flow acceleration at the inlet of the jet fans.

**Effect of the length of the 3D domain for the simulation of jet fans**

The location of the interface between the 1D and the 3D domains is fundamental for the correct calculation of the operative map of the jet fans. The flow field should be fully developed in the 3D domain (with a uniform flow over the cross section) before making the connection (i.e., interface) with the 1D domain. But at the same time, the 3D domain should be limited in length in order to reduce the computational cost of the simulation. A portion of the tunnel is simulated by imposing the pressure at the inlet and the velocity at the outlet. Two different velocities, i.e., 3 m/s and 6 m/s, are chosen because the decay length of the jet is affected by the velocity in the tunnel [7]. Higher values are not used because they are not common in ‘real-life’ applications. In this case, the tunnel is simulated without considering the friction losses at the walls, so the pressure distributions of the domains with different lengths can be compared together.

![Pressure along the tunnel for different domain’s lengths](image3.png)

The pressure distribution for the different simulations is presented in Figure 3, while the pressure difference at the portals is summarized in Table 2. From the comparison, the simulation with Length 2 is chosen because it ensures that the high speed jet induced by the jet fans is fully decayed. The operative map of the jet fans in the tunnel is evaluated using the set-up of Mesh 2 and Length 2. The pressure at the outlet of the tunnel is changed in order to draw the operative map of the jet fans in the tunnel. In order to avoid possible uncertainties due to
the friction losses the walls are modelled as free slip surfaces. The operative map of the jet fans is presented in Figure 4, together with the results of the Eq. 1 assuming an installation efficiency of 1.0.

Figure 4: Operative map of the jet fans.

Figure 5: Pressure comparison in cold flow.

Comparison between multiscale, 1D and 3D simulations

The tunnel is now simulated embedding the operative map of the jet fans in the 1D model. The results obtained with 3D, 1D and multiscale simulations are compared together, Figure 5. For sake of simplicity the portals are not modelled and the pressure is imposed at the ends of the tunnel. This simplification is required to avoid uncertainties over the local pressure losses occurring due to the section’s change. In the multiscale model only the first and the last 50 m of the tunnel are included. The results are presented in terms of pressure distribution along the tunnel and velocity (see Figure 5 and Table 3).

The results have a small error in the velocity, within 2%, so the model can be applied for the simulations of tunnels in cold conditions. The pressure rise induced by the fan is compared for the three models. The agreement between the 1D and the multiscale is better (with an absolute deviation of 0.2 Pa) than the one with the 3D simulation (with an absolute deviation of 2.5 Pa). However, the pressure rise induced by the fan is estimated by calculating the pressure losses along the tunnel, which might not be very accurate.

Table 3: Models’ comparison in cold conditions.

| Case      | Velocity | ΔPfan |
|-----------|----------|-------|
| 3D Sim.   | 6.30 m/s | 74.8 Pa |
| Multiscale| 6.17 m/s | 72.3 Pa |
| 1D Sim.   | 6.22 m/s | 72.1 Pa |

SIMULATIONS IN CASE OF FIRE

After the first simulations in cold conditions, the multiscale approach is tested with different fire loads and different lengths for the 3D domain. Also, in this case the tunnel is modelled without including the portals for sake of simplicity, but including a small 3D portion, 50 m long, near the ends of the tunnel. The fire is also simulated in a 3D domain, whose length varies as discussed later. The jet fans are not simulated in the 3D model but their operative map is embedded in the 1D network. Different fire loads are examined in order to show the applicability of the current model for the simulations of fire in tunnels.

Effect of the grid resolution.

The mesh sensitivity analysis is carried out with the same grid resolution tested previously, Table 4. The HRR in this case is 2.5 MW, which represents an intermediate value of the range of HRRs that will be simulated later, from 0.0 MW to 5.0 MW. A portion of tunnel, 100 m long, is simulated with a fire located at 20 m from the inlet portal. The total pressure, 20 Pa, is imposed at the inlet of the tunnel, while the static pressure, 0 Pa, is imposed at the outlet. In this case the pressure is imposed instead of the velocity because the latter numerical set-up yields some numerical issues occurring when a fire in a tunnel is simulated with FDS [9]. The temperature and pressure profiles are compared as a function of the grid resolution, Figure 6 and Figure 7.
The longitudinal velocity upstream the fire for the given pressure difference at the portals is given in Table 4, showing that Mesh 2 and 3 predict the same longitudinal velocity. The temperature profiles along the tunnel are similar with the 3 grids. There are though some discrepancies in the fire region, which is out of the scope of the current study. The pressure shows an unexpected behavior for Mesh 3: the pressure gradient between the inlet portal and the fire is positive, while it is negative for Mesh 1 and 2. With a positive pressure gradient a negative velocity is expected, but the pressure distribution obtained with Mesh 3 is a consequence of the large velocity fluctuations in the fire region. The pressure profile obtained with the finest mesh is not considered to be realistic. Therefore, the Mesh 2 is chosen for the next simulations where the length of the domain is investigated and the results are compared with a full 3D calculation.

**Effect of the interface location between 1D and 3D domains**

After comparing the results obtained with different grid resolutions, the size of the 3D domain is changed and the results are compared together for different values of the HRR, Table 5.

The pressure distribution along the tunnel changes with the HRR, for low values of the HRR, 3.0 MW and below, the pressure decreases along the tunnel except for the region downstream the jet fans and near the fire section. When the HRR increases, 4.0 MW or higher, the pressure in the 3D simulation has a positive gradient in the region downstream the fire, where a negative pressure gradient is expected, Figure 9. The same effect is present, but less visible, also in the multiscale simulations, where the pressure rises in the fire section. This effect becomes more evident when the HRR increases, because the flow accelerations in the fire region have larger intensity. In a fully 3D simulation, even if the pressure distribution changes with the mesh resolution, the velocity upstream the fire is weakly affected, the difference between Mesh 2 and 3 was negligible. In the multiscale, due to the interfacing of different models, this is generally not true because the central domain of
the tunnel is connected to the portals via 1D branches. The velocity along the tunnel have different trends increasing the HRR, showing that the multiscale is applicable only for relatively small fires, below 3.0 MW.

Table 5: Comparison of the longitudinal velocity for different domains’ size and HRR.

| Case               | Length 3D | Vel. 1.0 MW | Vel. 2.0 MW | Vel. 3.0 MW | Vel. 4.0 MW | Vel. 5.0 MW |
|--------------------|-----------|-------------|-------------|-------------|-------------|-------------|
| 3D Sim.            | 500 m     | 5.88 m/s    | 5.79 m/s    | 5.76 m/s    | 5.71 m/s    | 5.66 m/s    |
| Multiscale 1       | 100 m     | 5.85 m/s    | 5.77 m/s    | 5.69 m/s    | 5.60 m/s    | 5.53 m/s    |
| Multiscale 2       | 150 m     | 5.85 m/s    | 5.77 m/s    | 5.67 m/s    | 5.55 m/s    | 5.36 m/s    |
| Multiscale 3       | 250 m     | 5.90 m/s    | 5.83 m/s    | 5.73 m/s    | 5.61 m/s    | 5.45 m/s    |

CONCLUSIONS

The current paper describes the implementation of the multiscale simulation approach for tunnels in FDS. The tunnel was first simulated in cold conditions, without fire. A preliminary grid and length sensitivity study defined the appropriate numerical setup for the simulation of the jet fans in the tunnel. Based on this set up, the operative map of the jet fan was calculated with FDS and it was later embedded in the 1D model. When the tunnel is simulated in cold flow conditions, the velocity and the pressure distribution obtained with the different models are in good agreement. The pressure along the tunnel has a linear distribution as expected due to the friction losses along the walls. The comparison of 3D, 1D and multiscale simulations shows a satisfactory agreement between the numerical approaches. This result confirms the possibility of using FDS in a multiscale configuration to simulate the ventilation in cold conditions in tunnels.

The simulation of a fire in the tunnel induces an unexpected pressure distribution which is caused by the large flow accelerations in the fire region. The mesh sensitivity analysis shows that with a fine mesh, the pressure gradient is positive also in zones of the tunnel where the pressure is expected to decrease because of the friction losses. Therefore, a coarser mesh was chosen since the results of the two grids were similar in terms of velocity and temperature along the tunnel. When the fire HRR increases, some zones with positive pressure gradient appear also with the intermediate mesh. The level of agreement between the results obtained with the multiscale and with the 3D approaches decreases increasing the HRR. The current comparison shows that the pressure distribution along a tunnel is a key parameter for the feasibility of the multiscale approach. This is applicable only if the 3D and the 1D models can predict the same pressure distribution in the tunnel. This limits the possibility to currently use the multiscale approach for the simulations of fires in tunnels to HRRs below 3MW.

REFERENCES

[1] Ang, C. E., Rein, G., Peiro, J., & Harrison, R. (2016). Simulating longitudinal ventilation flows in long tunnels: Comparison of full CFD and multi-scale modelling approaches in FDS6. Tunnelling and Underground Space Technology, 119-125.

[2] Colella, F. (2010). Multiscale modelling of tunnel ventilation flows and fires. PhD thesis.

[3] Vermessi, I., Rein, G., Colella, F., Valkvist, M., & Jomaas, g. (2017). Reducing the computational requirements for simulating tunnel fires by combining multiscale modelling and multiple processor calculation. Tunnelling and Underground Space Technology, 146-153.

[4] Prince, J., (2015) "Coupled 1D-3D simulation of flow in subway transit networks.", PhD thesis.

[5] Tarada, F., & Brandt, R. (2009). Impulse ventilation for tunnels–a state of the art review. In Proceedings of 13th International Symposium on Aerodynamics and Ventilation of Vehicle Tunnels. BHRG. New Brunswick, New Jersey, USA (pp. 95-107).

[6] Beyer, M., Sturm, P. J., Saurwein, M., & Bacher, M. (2016, May). EVALUATION OF JET FAN PERFORMANCE IN TUNNELS. In 8th International Conference Tunnel Safety and Ventilation.

[7] McGrattan, K. B., McDermott, R. J., Weinschenk, C. G., & Forney, G. P. (2017). Fire dynamics simulator, technical reference guide. NIST Special Publication SP-1018.

[8] Li, Y. Z., Lei, B., & Ingason, H. (2010). Study of critical velocity and backlayering length in longitudinally ventilated tunnel fires. Fire Safety Journal, 45(6-8), 361-370.

[9] FDS user’s group (2017), Discussion over ‘Pressure rise induced by fires in tunnels’ https://groups.google.com/d/topic/fds-smw/H5BL3j9j4aM/discussion