Tunnelling and Underground Space Technology 64 (2017) 146–153

Contents lists available at ScienceDirect

Tunnelling and Underground Space Technology

journal homepage: www.elsevier.com/locate/tust

Reducing the computational requirements for simulating tunnel fires by combining multiscale modelling and multiple processor calculation

Izabella Vermesi [a,*], Guillermo Rein [b], Francesco Colella [c], Morten Valkvist [d], Grunde Jomaas [a]

[a] Department of Civil Engineering, Technical University of Denmark, 2800 Kongens Lyngby, Denmark
[b] Department of Mechanical Engineering, Imperial College London, SW7 2AZ London, UK
[c] Exponent Inc., 9 Strathmore Rd, Natick, MA, USA
[d] Greater Copenhagen Fire Department, Bag Rådhuset 3, 1550 Copenhagen W, Denmark

ARTICLE INFO

Article history:
Received 9 December 2015
Received in revised form 4 December 2016
Accepted 30 December 2016

Keywords:
CFD
Multiscale modelling
Tunnel fires
FDS6
Computational efficiency

ABSTRACT

Multiscale modelling of tunnel fires that uses a coupled 3D (fire area) and 1D (the rest of the tunnel) model is seen as the solution to the numerical problem of the large domains associated with long tunnels. The present study demonstrates the feasibility of the implementation of this method in FDS version 6.0, a widely used fire-specific, open source CFD software. Furthermore, it compares the reduction in simulation time given by multiscale modelling with the one given by the use of multiple processor calculation. This was done using a 1200 m long tunnel with a rectangular cross-section as a demonstration case. The multiscale implementation consisted of placing a 30 MW fire in the centre of a 400 m long 3D domain, along with two 400 m long 1D ducts on each side of it, that were again bounded by two nodes each. A fixed volume flow was defined in the upstream duct and the two models were coupled directly. The feasibility analysis showed a difference of only 2% in temperature results from the published reference work that was performed with Ansys Fluent (Colella et al., 2010). The reduction in simulation time was significantly larger when using multiscale modelling than when performing multiple processor calculation (97% faster when using a single mesh and multiscale modelling; only 46% faster when using the full tunnel and multiple meshes). In summary, it was found that multiscale modelling with FDS v.6.0 is feasible, and the combination of multiple meshes and multiscale modelling was established as the most efficient method for reduction of the calculation times while still maintaining accurate results. Still, some unphysical flow oscillations were predicted by FDS v.6.0 and such results must be treated carefully.

1. Introduction

Fire modelling is frequently used as a means of investigating a variety of fire scenarios. Whereas such modelling is feasible for most type of structures, long tunnels are problematic from a numerical point of view, because they, due to their length, are defined by large domains that require very large computational resources. This numerical challenge is amplified by the fact that finding the proper fire safety strategy often requires trying a number of scenarios to establish all the essential characteristics of the system. Among the models found in literature, Vega et al. (2008) required 50 h to simulate 10 min of a fire using a 3D model in ANSYS Fluent. Other models use either short tunnels (Jain et al., 2008) or simplified one-dimensional models for longer tunnels (Migoya et al., 2009). As an alternative to sacrificing either complexity or time efficiency, multiscale modelling for tunnel flows and fires has previously been studied using a general purpose computational fluid dynamics (CFD) software and it has yielded satisfactory results in comparison to full scale CFD simulations (Colella et al., 2010). The method combines a 3D domain for the near fire zones, which are characterized by large temperature and pressure gradients, with a 1D network approach for the bulk flow in the far field. A full description of the concept, as well as an assessment of the method's accuracy, is presented by Colella et al. (2010, 2011). The model used in Colella's work has been validated using experimental measurements from real tunnel flows (Colella et al., 2010). As the previous study used ANSYS Fluent, there is a need for a feasibility study using other modelling techniques.

Herein, the primary investigation is thus the feasibility of multiscale modelling of tunnel fires in Fire Dynamics Simulator (FDS v.6.0), which is a CFD model for fire driven flow that is widely used

---

* Corresponding author at: Department of Mechanical Engineering, Imperial College London, UK.

E-mail address: i.vermesi14@imperial.ac.uk (I. Vermesi).

http://dx.doi.org/10.1016/j.tust.2016.12.016
0886-7798/© 2017 Elsevier Ltd. All rights reserved.

for heat and smoke transport from fires (Jain et al., 2008). It is worth noting the difference in the governing equations in ANSYS Fluent and FDS: the former uses Reynolds-averaged Navier Stokes equations (RANS) for its simulations, whereas the latter uses Large Eddy Simulations. A comprehensive analysis of the differences between RANS and LES modelling are found in the work by Versteeg and Malalasekera (2007). Furthermore, FDS is an open-source software, thus it is very widely used, especially in the industry. Therefore, developing a feasible multiscale model for this software can make a significant impact on reducing simulation times for a large number of users. Parametric studies are needed in order to obtain the relevant scenario that is going to be analyzed in detail for the final tunnel ventilation design. This is possibly performed with more computationally heavy programs or settings. As such, the fast computation enables the designer to eliminate several scenarios in the process towards the final design.

The implementation of the multiscale model in FDS v.6.0, sketched in Fig. 1, followed the work by Colella et al. (2010), while taking advantage of the fire-specific capabilities of FDS v.6.0. The geometry and model guidelines such as the domain length and volumetric flow induced by the ventilation system served as a start for the present model. The 3D component was created using the traditional 3D grid, while the 1D network was implemented using the Heating, Ventilation and Air Conditioning (HVAC) feature of FDS v.6.0, which is described in detail in the next section. These two models are coupled directly in the FDS code, something which provides a continuous interaction between them. This is in contrast with an indirect coupling that requires the use of an additional software to model the 1D network, whose results are then used as input for the boundary conditions of the 3D model. As a result of this direct coupling, the time spent preparing the model is reduced. The coupling of 3D and HVAC component in FDS has been validated against real tunnel flow data in Ang et al. (2016), thus confirming the possibility of coupling the two domains.

The second objective of this paper is to assess which of the following methods is the most time-efficient: multiscale modelling, multiple processor calculations or a combination of the two. Because of the computational requirements, the 3D grid was divided in multiple meshes assigned to individual cores of the computer. In this way, the order of magnitude of the duration of the simulation decreased from weeks to hours. To find out which method had contributed the most to the decrease in runtime duration, simulations were performed on the same model, first using a single mesh in the 3D domain of the multiscale model to compare with the multiscale-multiple-mesh model. Then the full tunnel was simulated with a single mesh and multiple meshes.

2. Methodology

2.1. 3D model and fire scenario

In order to be able to compare the results of the method implemented in FDS with the results obtained in the reference work (Colella et al., 2010), the tunnel chosen for the analysis has a total length of 1200 m and a longitudinal ventilation system. It is considered to be a road tunnel with traffic going in one direction only. In the previous work, the tunnel cross-section had a horseshoe shape with a height of 6.5 m and a cross-sectional area of 53 m^2. However, FDS has some constraints regarding geometry and can only contain models with rectangular grids. An attempt was made to simulate a circular cross section using a stair-stepped boundary condition. However this is not a correct solution as it offers a different behaviour of the flow, which is not necessarily a realistic behaviour. Therefore, an equivalent, rectangular cross-section was used in the current study. In order to obtain an area similar to the previous one, the width was chosen to be 8 m, giving a cross-sectional area of 52 m^2. The hydraulic diameter is found to be $D_H = \frac{4A}{P} = 7.17$ m. The walls, floor and surface were defined as adiabatic concrete surfaces and the inlet and outlet connected to the HVAC solver were defined as HVAC surfaces in order to permit the interaction between the two models. The tunnel walls were assumed as adiabatic for the sake of simplicity. Other heat transfer boundary conditions to the walls could have been used, but the adiabatic condition gives conservative estimates of the pressure losses (highest fire throttling effect), back-layering velocity and back-layering distance (Colella et al., 2010). Furthermore, the missing heat loss in the 1D part of the tunnel results in increased temperatures and hence pressure losses, both of which in turn produce a conservative estimate of the backlayering distance due to a reduction in the airflow. Given that the main objective of the study is to evaluate the performance of multiscale modelling in FDS v.6.0 compared to another CFD code, it is deemed acceptable to use adiabatic conditions.

In order for the simulation to give accurate results, the boundary interface between the two models has to be placed at a location where the flow is fully developed and the temperature or velocity gradients are insignificant (Colella et al., 2012). As shown by Colella et al. (2011), accurate results are obtained when the distance from the fire is at least 13 times larger than the hydraulic diameter. Thus, a domain larger than 200 m should yield satisfactory results. Results recorded 10 m from the fire become boundary independent for grids larger than 200 m, whereas results recorded 100 m from the fire do not depend on the boundary interface for grids larger than 400 m (Colella et al., 2010). Therefore, as a starting point, the CFD domain was chosen to be 400 m along the longitudinal axis and the fire was placed in the centre, as is shown in Fig. 2.

The fire used herein has a maximum heat release rate (HRR) of 30 MW, which can represent the peak heat release rate for a burning bus (Carvel and Beard, 2005). The fire is represented using a lumped species approach for species tracking combined with a mixing model and fast chemistry. The radiative fraction, which is the amount of energy released from the fire as thermal radiation, was chosen as the default radiative fraction in FDS, namely 0.35. After an initial study of the fire development and change in mass flow rate with respect to time (Colella et al., 2011), the simulation duration was chosen as 600 s, at which point it was verified that steady-state conditions had been reached throughout the tunnel.

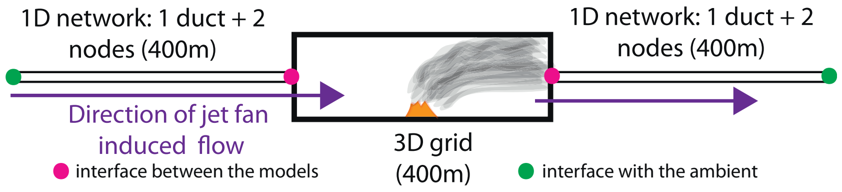

Fig. 1. The concept of multiscale modelling of tunnels fires in FDS v.6.0.