



# Two-scale Modelling Approach For Simulating Flows in Tunnel Fires

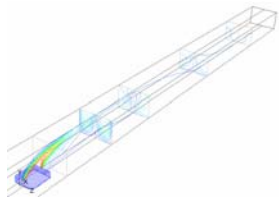
F. Colella<sup>1</sup>, R. Borchellini<sup>1</sup>, R. Carvel<sup>2</sup>, G. Rein<sup>2</sup>, J.L. Torero<sup>2</sup>, V. Verda<sup>1</sup>

<sup>1</sup> Politecnico di Torino – Dip. di Energetica, Italy

<sup>2</sup> University of Edinburgh, BRE Centre for Fire Safety Engineering, UK

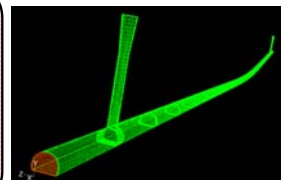


Politecnico di Torino  
 Department of Energetics  
 BRE Centre for Fire Safety Engineering



**Introduction**

The design of the ventilation system and its management are complex issues. The ventilation flow within a tunnel is dependent on the combined influence of ventilation devices (jet fans, axial fans), tunnel layout, slopes, the actual flow conditions at the tunnel portals, the presence of obstacles or vehicles as well as the effect of the fire size and fire location. This work develops a novel modelling technique for flow conditions in case of fire, coupling the plume with the tunnel geometry and the ventilation system



**Modelling techniques**

**Mono-dimensional Model**

The most practical approach to model the tunnel and its ventilation systems is to use mono-dimensional thermo-fluid dynamics models. This approach is the only one which can deal with a global representation of the system including tunnel, shafts as well as fans. The intricate network (Fig. 1) of ducts and the resulting flows and temperatures can be only solved efficiently using this method

Fig.1. Example of network layout – the computational domain is splitted in branches and nodes. Continuity and momentum equations are written in mono-dimensional fashion

- Fast simulations for the whole tunnel domain
- Used for design purposes
- Need calibration constants
- Not able to represent smoke stratification or plumes

**CFD model**

CFD models are the only tools able to provide reliable data when specific characteristics of the flow field need to be known but they suffer from the difficulties in modelling turbulence combustion and radiation. CFD technique applied to tunnel fire suffers from the limitation induced by the unknown Fire size and growth, Vehicles arrangement and by the Huge computational Domain

- Detailed flow field representation
- Used for design verification
- Huge computational time
- Not affordable for long tunnels or for analysis of ventilation strategies

Fig.2. Example of flow field generated by operating jet fans

CFD models are usually applied to a small part of the domain where the ventilation conditions are considered as known. This produces an uncoupling between fire and ventilation system

**Multiscale techniques**

Fig.3. Typical temperature contours for tunnel fires

Fig.4. Typical velocity contours in presence of jet fan

The presence of **fire** or **fans**, produces fluid regions characterized by high temperature and velocity gradients (Fig. 3 and Fig. 4). These disturbances last for a limited small distance within the tunnel

Flow regions characterized by **low temperature/velocity gradients** can be successfully modelled using **mono-dimensional models**

Flow regions characterized by **high temperature/velocity gradients** are previously modelled using **three dimensional CFD models** and then their behaviours represented in terms of bulk flow quantities enabling a **coupling with the mono-dimensional model**

- Dramatic reduction of the computational time
- More realistic representation of the flow field using 1D models
- Successfully used for parametric studies on many scenarios
- The coupling between fire and ventilation is maintained

**A case study**

Fig.4. Tunnel layout – 14 pairs of jet fans located close to the portals. Fire located in the centre with sizes ranging from 10 MW to 50 MW

CFD model of the fire region has been developed and its behaviour investigated under different pressure differences obtaining characteristic curves

Fig.5. Fire characteristic curves – the curves correlate bulk mass flow rate and pressure gradient across the fire zone. They are parameterized as function of the fire HRR. The results are obtained using the CFD code FLUENT 6.3.26 .

**Fire Characteristics**

- Sizes from 10MW to 50MW
- Modelled as rectangular obstruction releasing hot combustion products from the top and extracting cold air from the sides (Fig. 6)
- The dimensions of the obstruction have been scaled for different fire sizes using Froude scaling.  $\Phi^* = \frac{\Phi}{\rho_a c_p T_a D^2 \sqrt{g D}}$

Fig. 6. Fire source geometry

The curves introduced in the mono-dimensional model allow a more realistic description of the fluid-dynamics taking into account the pressure difference generated by the fire and its dependence on the fire size

- The flow resistance and buoyancy of the fire is not negligible
- The no fire calculations underestimate the ventilation requirements by up to 40%

Fire size	fans upstream the fire				fans downstream the fire			
	# min fans pairs	vel [m/s]	# min fans pairs	vel [m/s]	# min fans pairs	vel [m/s]	# min fans pairs	vel [m/s]
10MW	3	2,84	4	2,97	4	3,18	4	2,86
30MW	6	3,13	7	2,9	7	3,03	8	2,82
50MW	10	3,22	14	3,25	12	3,33	14	3,25

Table 1. Minimum ventilation requirements to achieve critical velocity – Comparison between 1D model results and 1D+3D model results.