Imperial College London
Department of Mechanical Engineering

Engineering simulations in the design and analysis of tunnels for fire safety and smoke ventilation

Chin Ding ANG

Supervised by Professors Guillermo Rein and Joaquim Peiró

Submitted in part fulfilment of the requirements for the degree of Doctor of Philosophy in Mechanical Engineering 28 March 2024

Declaration of Originality

I, Chin Ding ANG, declare that this thesis and the work described within have been completed solely by myself under the supervision of Professors Guillermo Rein and Joaquim Peiró. Where others have contributed or other sources are quoted, full references are given.

Abstract

In the design of tunnel ventilation systems, ranging from transport to utilities tunnels, engineering simulations are used by engineers globally. The simulations include 1D model and 3D computational fluid dynamics (CFD). CFD is popular in fire safety engineering, and the Fire Dynamics Simulator (FDS) is widely used. This thesis sets out to answer one question: Are engineering simulations the right tools for tunnels fire modelling? The short answer is yes, with a longer answer in this thesis.

Carbon monoxide, or CO is the principal culprit in fire deaths, and is an important design parameter. The research shows when considering CO production and simulation, a range of CO yield should be considered. This is due to the inherent variability with CO production in experiments. Secondly, this thesis demonstrates because engineering simulations are a simplified reality, model validation is critical. With observations of oscillation when modelling large fires in a longitudinally ventilated tunnel, this thesis shows there is a significant oscillatory behaviour for fire sizes over 30 MW. This resulted in identifying numerical issues in FDS, and contributed to FDS developers fixing this. Using engineering simulations, this thesis shows a 1D model remains important in design by using a simplified 1D model developed in this thesis, TE1D, to describe fire throttling in tunnels. The effects of fire throttling are assessed, that is the additional pressure drops in the tunnel due to a fire, occurring at the fire's location, and downstream of the fire. This thesis resulted in a novel quantifiable model that permits this effect to be calculated in design. This was not the case with previous models.

1D and 3D engineering simulations remain critical in design. This thesis concludes engineers need to understand the right time to use the right tool, and the limitations of these tools.

Acknowledgements

I would like to express my sincere thanks and appreciation (in no particular order) to:

- My PhD supervisors Professors Guillermo Rein and Joaquim Peiró
- My families including my father, mother and my sisters for their unrelenting support
- My second family, that is Hazelab Imperial College London
- Dr Roger Harrison for his contribution in my Masters, which formed the foundation to my work here
- Kuan Lin for his initial exploration in understanding fire throttling. The preliminary explorations of the ideas presented in Chapter 5 is part of his final year project [1]
- Dr Ingo Riess for his wonderful collaboration in our work on fire throttling
- The FDS developers including Dr Randy McDermott, Dr Kevin McGrattan and Dr Jason Floyd in their support for our work in understanding fire induced oscillations
- Dr Jonathan Barnett in our early collaboration to understand the modelling of carbon monoxide and its uncertainties
- Gordon Garrad for nurturing my fascination in research, and his feedback on some of the papers we published
- Ray Christie for teaching me so many rail and tunnel related issues
- Geri Mills (Martin) for collaborating with me on a technical paper that contributed to my thesis.

Finally I like to express a sincere thank you to everyone I have crossed paths with in developing my work. Thank you again.

Dedication

I first began my journey in research through my research project when completing my Master's at Imperial College London in 2014. Before that, ever since starting my engineering career in 2006, I have always been fascinated by the 'why' behind published design guidance.

It is through my engineering work that my fascination continued to grow in trying to understand the research that underpins how engineering is delivered. It fascinated me, but also dawned upon me that the design guidelines we use today can be traced back to research in 1950s and earlier.

It was from here that my aspiration grew where I wanted to better bridge the knowledge sharing between engineering practice and research. This firstly culminated in me completing a Masters at Imperial College London where I first met both my current PhD supervisors, Professors Guillermo Rein and Joaquim Peiró. They have been with me on my journey since 2013, and I am forever thankful for their support in helping me grow. I could not have finished my thesis and research without their support.

I would like to dedicate my thesis to Gordon Garrad. Gordon was my manager in my first job as a fire safety engineer and he cultivated my fascination in understanding the first principles behind design guidelines. It was through Gordon that I continued to develop my interest in research, and the interest blossomed under the tutelage of Professors Rein and Peiró.

Finally and importantly, I want to dedicate this to my families, for without their support and patience I could not have completed my thesis. It is because all of you had my back that I am able to do what I do.

‘If I have seen further, it is by standing on the shoulders of giants.’

Isaac Newton

Contents

List of Figures

Chapter 1

Introduction

1.1 Fire Safety Design in Tunnels

Tunnels are a marvel of engineering that have enabled the underground transportation of people, trains, vehicles, freight, water and utilities. The focus of our [1] thesis is on fire safety within transportation (roads and rails) tunnels.

Fires in these tunnels are low probability events. Australian statistic [11] suggests 1 vehicle fire per 130 million vehicle kilometres travelled. To illustrate, approximately 80,000 cars per day travel through Sydney Harbour Tunnel [12], one of Sydney's busiest road tunnels, and a vehicle fire is expected once every ten years.

Although the majority of these fires are controlled and do not escalate into fires that could threaten a life, there have nonetheless been tragic tunnel fires globally that led to the loss of lives. In the last two decades, there have been tragic fires [13] such as the 2006 Viamala Tunnel (Switzerland) or 2007 Burnley Tunnel (Australia) where 9 and 3 lives were tragically lost.

This means although modern tunnels have a reasonable track record when compared to building fires, fatalities are still occurring in these low probabilities but with catastrophically life

[1] The work contained in this thesis is my own work. I have made the conscious decision to use 'our' or 'we' throughout the thesis to reflect the collaborative nature of engineering design and this research in particular. This thesis is my own work, and the ultimate responsibility is mine.

changing consequences events.

Thanks to consistent public fire safety messages, society at large now understands the number one cause of fire death is smoke inhalation coupled with exposure to high temperature.

What is uncommon to society at large, but well known to engineers and regulators is that a tunnel ventilation system is provided not only to remove vehicle fumes, but more importantly to provide ventilation to reduce the impact of smoke and heat in a tunnel fire. This means an incorrectly designed or operated tunnel ventilation solution can result in injuries, and fatalities.

In the fire safety design of tunnels, the ventilation system form an integral part of the tunnel fire systems to safeguard the life of the motorists or passengers (rail) in the tunnel. For a tunnel ventilation system, the fundamental principle of the system is simple - the key for fire safety design in tunnel is to push enough air along the tunnel so that smoke and heat from a fire can be sufficiently diluted to minimise the harm to any occupants who may be present in the tunnel.

As such, the focus of the tunnel ventilation system is to ensure the air flow required is adequate, and to understand how to predict the impact of a fire in the ventilation system.

With the advancement in engineering technologies over the years , the design of tunnels has evolved from desktop calculations and 1D (one dimensional) models, to 3D computational fluid dynamics (CFD) models.

1D and CFD modelling are now widely used tools in the design of the tunnel ventilation system in these tunnels, and collectively these are referred to as engineering simulations - the focus of this thesis.

1.2 Engineering Simulations in Fire Safety Design

1.2.1 Overview of Tunnel Ventilation System

A tunnel ventilation system is provided to remove heat and fumes, for example from road vehicles, from a tunnel. In addition, the system is a main fire safety system in a tunnel that is used to remove and reduce the impact of heat and smoke from a fire. Across the world, the majority of newer transport tunnels over a few hundred meters are normally provided with a tunnel ventilation system.

There are three main types of tunnel ventilation: longitudinal, transverse and semi transverse. See figure 1.1. A longitudinal system is commonly adopted due to its simplicity in the design, construction, operation and maintenance of the system. This is because a transverse or semi transverse system requires an additional ventilation plenum or duct, and this increases the complexities of such systems. The focus of this thesis is on the longitudinal tunnel ventilation system.

For the fire safety design in a tunnel, the purpose of the ventilation system in a fire emergency is to control the heat and smoke such that occupants in the tunnel can evacuate, and the fire brigade is able to enter the tunnel. Briefly, the design process for a tunnel ventilation system for this purposes includes the following considerations:

- **Determine the tunnel ventilation system:** The focus is on a longitudinal system. Its design is based on pushing air in one direction, normally in the direction of the traffic.

  In a fire, the system would aim to minimise smoke flowing upstream, i.e. against the ventilation direction. This creates an area relatively free of smoke to allow occupants to evacuate, and to allow fire brigade to approach the fire from clear air. The ventilation downstream of the fire also serves to dilute the smoke, reducing its concentration, and to lower the temperature of the smoke.

- **Determine the size of the fire:** The size of the fire dictates the capacity required for the ventilation system. Typically various fire sizes are selected to reflect the types of vehicles expected in a tunnel. For example, for a typical non-electric vehicle passenger car, the fire size is 5 MW [5], which is comparable to a fire involving five single seat sofas. A typical ladened heavy goods vehicle is approximately 50 MW, or the equivalent of 10 passenger cars on fire, to allow for some variability in the goods.

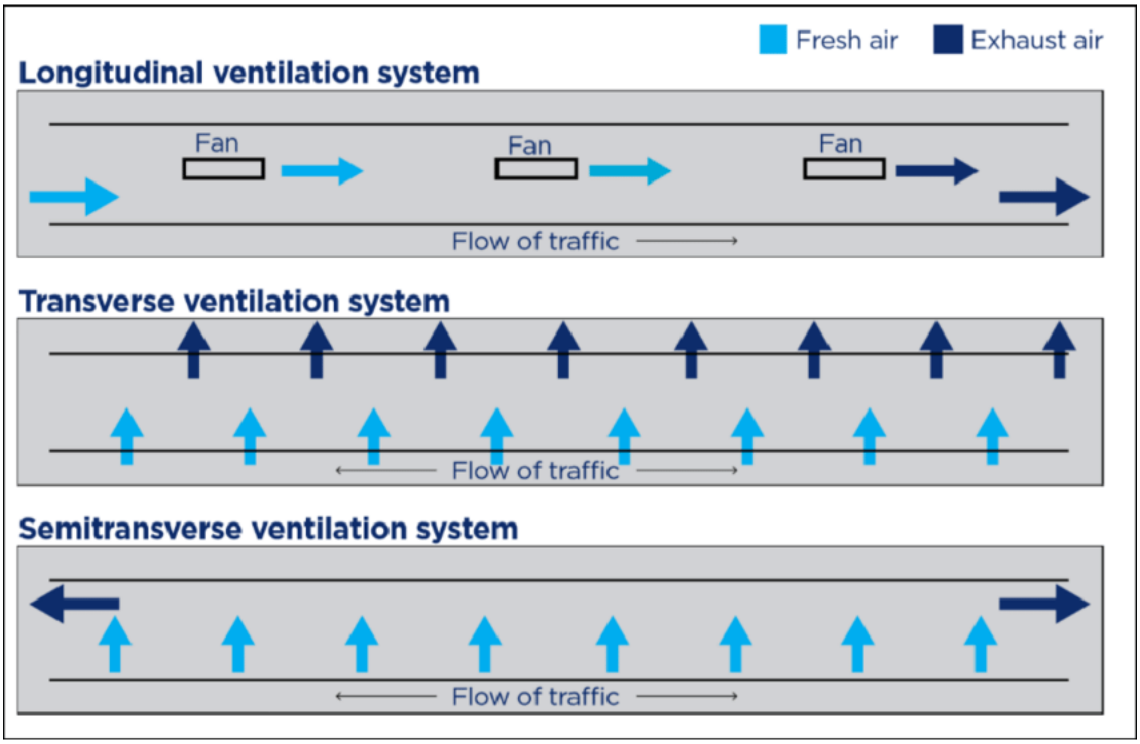


Figure 1.1: Three main ventilation systems for road tunnels [2]

- **Determine the evacuation route and number of occupants evacuating:** This is to determine the time it takes to evacuate the occupants, and it is used as part of the fire safety engineering design.

- **Engineering analysis for the ventilation system:** Once the fire size and the ventilation system is determined, preliminary calculations are normally done using a 1D or 2D method to estimate the flow required for the system. This flow is then used to size the tunnel fans, which in turn is used to spaceproof the tunnel, i.e. making sure everything fits, and to ensure there is adequate power supply. This simplified calculation method is fast, and is normally done at the earlier stages of a project.

  Following on that, a more involved CFD analysis is carried out to understand the interaction of the ventilation system and the fire. The results of the CFD analysis are used in conjunction with the evacuation time to establish if changes or modifications are needed for the ventilation system, or the egress route design.

This section discusses the key aspects of engineering simulations that are specifically relevant for ventilation and fire modelling in longitudinally ventilated tunnels.

1.2.2 CFD and Fire Dynamics Simulator

Computational fluid dynamics, or CFD, is commonly used in engineering design for fire safety and it has been used in the mainstream design process for the last 15 years. When CFD was first used in maintain fire safety engineering design, it has been largely used in larger, more complex buildings such as shopping centres. Today, CFD is used in projects of all shapes and sizes from a warehouse to a large international airport.

FDS, or Fire Dynamics Simulator [14] forms a major milestone in the increasing popularity of CFD in fire safety design. There are several key reasons for this popularity including its open sourced nature, purpose designed mathematical models for fire modelling and the availability of GUI (graphical user interface) to simplify the use of CFD.

FDS is used to model fire and smoke movement in a space. For the fire safety design in tunnel ventilation system, CFD is firstly used to show the tunnel ventilation system can achieve sufficient pressure to keep smoke in check. Secondly it is to show that the effluent from a fire, i.e. carbon monoxide (CO) and temperature, will not impact life.

A critical aspect of CFD, in particular for fire modelling which is a niche subject in the computational industry, is the verification and validation of the model. Briefly, verification and validation are important for CFD, and FDS because they provide help users assess wether the simulations reflect the physical reality.

Although the FDS developers and communities have compiled many fire experiments to be used as part of the verification and validation for FDS [15], even after 20 years where FDS was first publicly released, there are fewer than 50 experiments compiled, with only two experiments directly based on tunnels. These are documented in the FDS validate guide [15] where the guide is regularly updated.

Advancement in numerical fire modelling is critically dependent on the availability of good fire experiments, in particular for tunnel fire models due to the scarcity of such models. As such, this raises a question whether we, and by extension the tunnel fire communities can trust CFD and FDS.

The objective of this thesis is to examine the capabilities of FDS on aspects that matter when considering fire modelling in tunnels. The findings from this thesis will improve the understanding of engineers when designing and modelling ventilation system in tunnels.

We wanted to answer the question, CFD is very popular, with FDS being widely used in real world engineering design, but are CFD and FDS always the right tools for tunnel fire modelling?

We attempt to answer this question by considering the following:

1. Assess the limitations of Carbon Monoxide (CO) modelling, and how this limitation is driven by the inherent complexities of CO production. CO is an important variable when considering safety in tunnels.

2. Demonstrate the importance of pressure solver when modelling a fire in the tunnel through the observation of unexpected oscillation.

3. Understanding of fire throttling from first principles using a 1D model, and demonstrate through FDS the limitation of a 1D model.

These are further discussed in the following sections. Note FDS is the main CFD package used in this thesis and is featured throughout all chapters.

1.2.3 Carbon Monoxide (CO) Concentration

It is now widely understood that CO, or carbon monoxide, is major byproduct of fire is the main culprit in fatalities and injuries in fires. Therefore, in fire safety design, it is natural to attempt to simulate the CO concentration in a fire.

With the advent of CFD and FDS, engineers can now simulate the presence of CO concentration in fire. Because of this capability, CO concentration is commonly used today as a predictor of safety in tunnel fire safety design. This is achieved by modelling the fire and its impact using CFD and FDS. The simulations help engineers understand the sizing of our tunnel ventilation fans, and whether additional dilution (more air flow and therefore fans) is required.

Our research is to show the limitations of CO modelling, and how this limitation is driven by the inherent complexities of CO production. We show this limitation is fundamental, as even repeated experiments cannot reliably repeat the prediction of CO.

This is further discussed in Chapter 3, and we show in this chapter how FDS could be used to model CO prediction in a growing and steady state fire, but not in a decaying fire.

1.2.4 Unexpected Oscillation

With CFD and FDS, the use of the simulation is to verify the tunnel ventilation system design when a fire occurs in the tunnel. Therefore, on top of understanding the usage of engineering simulations and FDS in tunnel fire safety design (Chapter 2) and how simulations are constrained by the physical world (Chapter 3), the solver itself is another major focus when considering simulations for engineering application.

It is important to understand if the unexpected solver's behaviour, as we will describe briefly below, is physical or numerical.

In 2014 as we were working on exploring multi-scale modelling [16], we discovered an unexpected oscillation in the mass flow. This phenomenon was never reported, and when we first discovered the oscillation, we initially thought the oscillation was physical. Significant oscillations of the mass flow in the tunnel prevents us from obtaining a reliable estimate of the actual mass flow, and thus it impacts the assessment of the fan sizes we need to achieve a safe design.

We have spent time and additional modelling, as discussed in Chapter 4, to better understand the oscillation. Further, we have spent the last 6 years convincing the tunnel fire safety community, and the FDS developers that there is a numerical issue associated with modelling large fires in long enclosed tunnels using FDS. This issue was finally acknowledged and resolved in

2022 [17].

We show in this chapter our journey in discovering and understanding whether the unexpected oscillation is numerical or physical. Through the work here, we highlight the need, in any engineering simulations, to understand whether what is predicted by the model is physical or numerical, as failure to do so result in over sizing, or more importantly undersizing the fans required for tunnel fire ventilation.

1.2.5 Fire Throttling

Following previous work in understanding the limitations CFD simulations and FDS in particular, this chapter of the thesis aims to demonstrate two points.

Firstly to better understand fire throttling. Briefly, fire throttling is the additional pressure loss due to a fire in a longitudinally ventilated tunnel. This is discussed in greater detail later in Chapter 5, and suffice to say this is an important issue because at present, although the issue is recognised in design guidance, the understanding of this phenomenon remains fragmented.

Through the work in this chapter, we wanted to understand this phenomenon through simple first principles using a 1D model. Thereafter, we compared the 1D model to the simulations in both OpenFOAM and FDS.

Secondly to demonstrate that when it comes to engineering simulations, a simple 1D model can go a long way in understanding fundamental phenomenon and in tunnel ventilation design. We have accomplished this through a model we have developed, named TE1D (throttling effect 1D), that provides a simple 1D description for fire throttling based on pipe flow principles.

In addition, we also showed it is critical to understand the limitation of a model, in this case the 1D model as we showed the importance of CFD (or 3D simulations) because we showed for a large fire exceeding 30 MW, the 3D effect downstream of the fire becomes a driving factor for fire throttling.

At the end of this chapter, we have successfully provided a 1D model that can be adopted for practical application or engineering design. The model is based on general pipe flow principles, which is the fundamental principle used in tunnel ventilation design as a tunnel is essentially a big duct or pipe. This is a significant improvement over the status quo as the existing equations have not adequately described the effect of fire throttling downstream of the fire (the equations only describe throttling at the fire's location).

The work is presented in Chapter 5.

1.2.6 Objective of this Thesis

For this PhD research and thesis, we set out to answer a fundamental question:

Computational fluid dynamics (CFD) is very popular, with Fire Dynamics Simulator (FDS) being widely used in real world engineering design, but are CFD and FDS always the right tools for tunnel fire modelling?

1.3 Publications

The work contained in this thesis has been derived from materials published or presented in journals and conference papers, but it has been modified to make the thesis more coherent. The papers of which I am the lead author and where the work has appeared in this thesis are:

- Master's thesis [16] titled *Investigation of a computationally efficient multi-scale modelling method in long tunnels for FDS6.*

- Journal paper [3] titled *Simulating longitudinal ventilation flows in long tunnels: Comparison of full CFD and multi-scale modelling approaches in FDS6.*

- Conference paper [4] titled *Unidrectional ventilation in rail tunnels.*

- Journal paper [18] titled *Unexpected oscillations in fire modelling inside a long tunnel.*

- Conference paper [19] titled *Numerical prediction of CO and its impact on practical application.*

- Conference paper [20] titled *Understanding and modelling the fire throttling effect in longitudinally ventilated tunnels.*

- Journal paper [21] titled *Analysis of fire throttling in longitudinally ventilated tunnels with a one-dimensional model.*

Chapter 2

CFD and Fire Dynamics Simulator

2.1 FDS in Practical Application

This chapter introduces the main CFD package, Fire Dynamics Simulator Version 6 [14], used in the investigation carried out in this thesis. This chapter provides a general introduction to FDS including some of the key features from the perspective of an engineer.

FDS is an open source solver with a large, thriving community and this contributes to the popularity of FDS in research and practical application. Further, the FDS community has compiled over 40 fire experiments [15] that are used in the development of the code. This is part of the validation and verification activities for the solver and is discussed later in Section 2.4.

The objective of this chapter is to cover the main considerations for an engineer when using FDS to carry out tunnel fire models. There are issues unique to modelling tunnel fires using FDS. This chapter attempts to capture these main issues, but does not intend to describe in detail the mathematical models, or the various numerical methods although some aspects, such as verification and validation will be discussed where relevant. This is because there are other better descriptions for the numerical methods of FDS such as the FDS Mathematical Model guide [22].

This chapter also describes the general considerations we used when carrying out the CFD modelling in the subsequent chapters. The assumption of this chapter is readers have an appreciation of fire safety design, and of using FDS including setting up a model, running a model and post processing.

This chapter includes ideas developed from my previous work [3, 16].

2.2 A Brief Introduction to FDS

FDS, or Fire Dynamics Simulator, is an open source CFD package developed by NIST (National Institute of Standards and Technology) and VTT Technical Research Centre of Finland in 2000 [14]. FDS is developed to numerically model thermally driven, low Mach number flow represented by a modified form of the Navier-Stokes equations.

From experience in practical application, FDS has been predominantly used to model compartment (room) fires since it was first introduced. Over the last decade, increasingly FDS has also been used to model fires in tunnels.

As noted previously, FDS has a thriving community via Google Groups [23], with active participations from the developers and contributors. In addition to the main solver, FDS is packaged with a post processing and visualisation tool, Smokeview (SMV). The popularity of FDS is largely contributed by being an early pioneer. It was first publicly released in 2000 when CFD fire modelling was growing, its open source nature, and it has gained popilarity thanks to its specialisation in smoke and fire modelling and its thriving community of researchers and engineers.

In practical applications, the popularity of FDS is also helped by the availability of commercial GUI (graphical user interface) packages, such as PyroSim, that greatly simplify the model building process including the ability to import complex geometries via computer aided design (CAD) models.

A brief summary of FDS is included below. This is not intended as an in-depth explanation of

FDS or its mathematical models. Instead, the focus of this summary is on the main assumptions relevant to tunnel fire modelling.

Large Eddy Simulation, LES. LES is the main turbulence model in FDS. The reason for using LES is that for DNS (Direct Numerical Simulation) to be applicable, the model's grid resolution should be smaller than the Kolmogorov scale, η (length scale of the smallest turbulent eddies). For fire modelling, η is normally 1 mm, and for a tunnel model of 1 kilometre, there would be over 7 billion cells, which is well beyond the current limit for practical application at 10 million or less.

LES is used in FDS primarily because of the turbulence model of LES being suitable for the mixing reaction of the combustion, i.e. a real fire where the turbulence model would impact the heat and smoke generated, as opposed to a pre-mixed combustion.

A major consideration for LES, as opposed to other models like RANS is the heavy reliance on the cell sizes. The quality of the predictions for LES is critically dependent on the length scale of the filter, where length scale in the context of FDS is the grid size. This is because the turbulence models only solve the length scale specified, and assumes anything smaller is ignored. This means the quality of the results rely on selecting the right length scale (grid size). The challenges lie in an engineer being able to select the right length scale where if it is too coarse, the simulation may finish sooner but the results may be unreliable (does not reflect the physical world), or if too fine, the simulation may require significant computational time than practicable in practical application.

The above is one of the reasons FDS developers have compiled a series of validation studies [15] where users of FDS are encouraged to compare the simulation against the validation studies via a set of non-dimensionalised parameters to have some confidence in the prediction. On top of this, users of FDS should carry out a grid sensitivity check by modelling increasingly finer but practicable mesh size to check that the results have reached grid convergence.

One challenge in the tunnel fire modelling community is the lack of tunnel fire experiments that can be used to validate models. Throughout this thesis, we highlight a consistent plea from the tunnel fire modelling community asking for more, and more diverse tunnel fire experiments as these experiments contribute immensely to the development of better numerical techniques.

Mass and species transport. Six species (fuel, O_2, CO_2, H_2O, CO, N_2) plus soot particles are tracked in the mass transfer model. The fuel is tracked as a single specie and the air plus the other products are lumped into a mixed species that are tracked as a single lumped specie. This is a key feature of FDS and is useful from a practical application perspective as it enables users to model the presence of soot and asphyxiant in a fire.

Soot is an important consideration in engineering design as engineers need to understand the extent of reduction of visibility due to soot concentration in a fire as the reduction in visibility adversely impacts the wayfinding ability of occupants.

The ability to model asphyxiant such as CO in a fire is increasingly becoming a critical variable in design. This is further explained in Chapter 3, but briefly, CO is a major cause of fatalities in fires and the ability to model this in design can in theory assists designers to understand the tenability of an egress route, for example in a long tunnel where occupants may need to travel hundreds of meters to reach an exit. We explicitly note in theory here that the mass and species modelling in FDS is critically dependent on user input as the users need to determine the conversion rate (a variable known as yield) for these species, and this is the theme of Chapter 3 where we highlight the challenges associated with this.

The mass and species transport include the equations to represent the addition of mass from evaporating droplets and other sub-grid particles such as water or fuel spray. This is noted here only for interest and is not the focus of this thesis.

Low Mach number approximation. The low Mach number approximation is a major point of FDS. Low Mach number in this instance refers to velocity at approximately 100 m/s or less. This means FDS is not suited for models involving high velocity flow, for example pressure vessel fire.

The low Mach number approximation is also an interesting challenge when modelling tunnel fires, which is the focus of Chapter 4. This is because the one onsequence on the formulation of the low Mach number Navier-Stokes equations is that the speed of sound is infinite [17]. Simplistically, in a tunnel several kilometre long with only an opening at both ends, the turbulent nature of a fire and its plume fluctuating in time would result in the expansion of gas. This resultant expansion would propagate through the tunnel in the speed of sound (physical world) at approximately 300 m/s, yet FDS is designed for low Mach number. This discrepancy in the pressure propagation results in additional challenges modelling a tunnel fire, described in this chapter and in Chapter 4.

Combustion [22, 16]. By default FDS uses a mixing-limited, infinitely fast reaction of lumped species (see mass and species transport) as its combustion model.

To account for incomplete combustion, e.g. under-ventilated fires, FDS uses a two-condition extinction model that considers the auto ignition temperature of a fuel in a cell and by considering whether a mixture of air and fuel in a cell can raise its temperature above an empirically determined critical flame temperature. The extinction model in FDS, and in the larger combustion community, is still an area of research. This explains the developers' advice that FDS should not be used to model under-ventilated fires, and why this is an important parameter in the model's validation study. This also contributes to the complexities in modelling of CO as discussed in the mass and species transport, and in Chapter 3.

Thermal radiation [22, 16]. FDS has the function to model thermal radiation from a fire. This feature is not the focus of this thesis, and is only briefly introduced here. The thermal radiation is based on the energy equation, and the temperature influences the thermal radiation modelled.

Because of the inherent restriction on the grid sizes where approximately 1 mm is needed to model flame sheet (see also LES discussed earlier), the computed temperature in a cell for a fire model with grid sizes in metre would be significantly less than those expected of a flame sheet in an actual fire. This is the critical reason why FDS is unlikely feasible in practical application to be used to model direct flame impingement, for example on a structure. This is specifically noted here as there have been examples of projects where FDS simulations were relied upon to understand the impact of direct flame impingement on structures.

Numerical solution [22]. Simplistically, FDS uses a predictor-corrector scheme for the computation. At the predictor stage, the density, mass fraction of the species being modelled and the velocity vector for the next time step is estimated. Subsequently, the temperature, then velocity divergence and the Poisson equation are solved. This then allows the velocity for the next time step to be estimated.

Prior to advancing to the corrector stage, the time step, δt is checked to have satisfied the CFL stability condition where the CFL number is close to, i.e. 0.8 but not greater than 1. The procedure only continues to the corrector step if the CFL condition is satisfied. Otherwise, the predictor stage is repeated again with a 10% change in the time step. At the corrector stage, the procedure is similar to those in the predictor stage but with corrections applied to the computed values.

The above provides a summary to FDS on areas that are relevant to this thesis. More detailed information can be found in the FDS Technical Reference Guide [22] and hence not repeated here. The next section describes the key reasons why modelling a tunnel fire is different than a compartment (room) fire, and the three main points to consider.

2.3 Tunnel Fires and Compartment Fires

2.3.1 Overview

Unlike a fire model in a room where the fire typically can disperse in all directions, tunnel fire modelling presents a unique fire modelling problem. As shown in the figure 2.1 below, a tunnel is a confined space where the fire is often confined in tight spaces, and smoke from a fire typically travels extensively in the longitudinal direction.

As noted earlier, the solver in FDS is based on a low Mach number approximation of the Navier-Stokes equations. A key feature of the low Mach number approximation is the assumption that the flow is incompressible thus pressure changes propagate instantaneously. Although this may

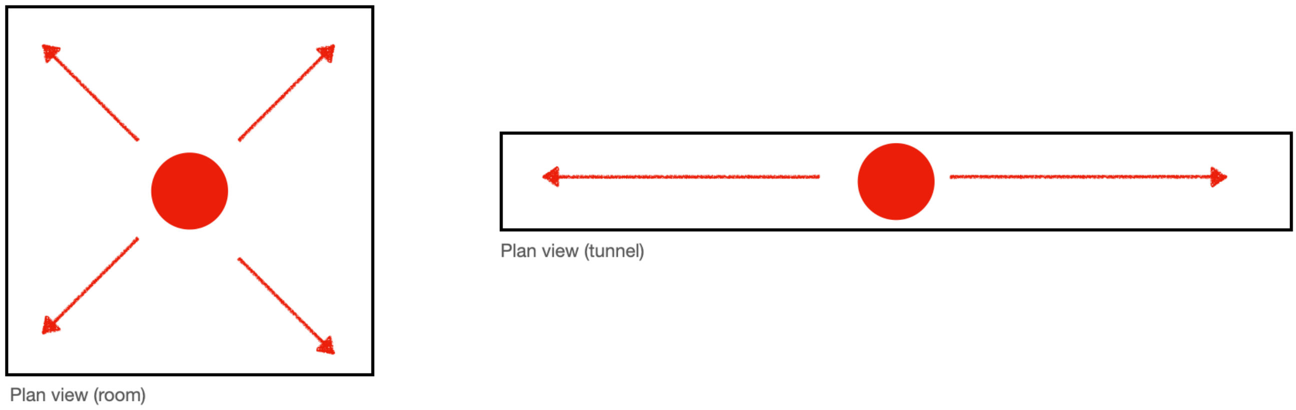


Figure 2.1: Plan view of the smoke and heat dispersion for a fire in a room (left) and fire in a tunnel (right)

be reasonable in a square room where smoke and heat from a fire can propagate in all directions, this is not the case for a long tunnel.

For example, in a 2 km longitudinally ventilated tunnel, the pressure changes generated by the jet fans at one end of the tunnel would take tens of seconds to reach the opposite end of the tunnel. This issue presents a unique challenge to modelling tunnel fires, and it has been a major issue we have brought to the attention of the FDS community [18], and that has finally been acknowledged and resolved by the developers [17] since this issue was first raised in 2016. This issue is further discussed in Chapter 4.

From a fire modelling perspective, all the important interactions and computations occur near the fire, and the further you are away from the fire, the simpler the interactions become.

There are three key points unique to modelling a fire in a tunnel that do not occur when modelling a compartment or a building fire. The main cause for these key points are due to the inherent characteristics of tunnels that are round, or circular (recall FDS is based on a rectilinear grid) that span across thousands of metres. As engineers will know, the footprint of most buildings are tens to a couple hundred metres typically.

These reasons result in unique issues when modelling a fire in a tunnel including:

- **Simplified geometry, including duct or rectilinear block arrangement.** This is an issue that impacts solvers such as FDS with rectilinear grid. This simplified geometry means engineers need to decide how best to represent a circular tunnel that curves, and with a varying degree of slope.

- **Tolerances of the pressure solver in FDS.** This issue is discussed earlier, and in greater detail in Chapter 4. The main issue of this is due to the low Mach number set up of the model, see Section 2.2.

- **Truncation of the tunnel's length.** This issue is due to the limitation of computational resources and the computational domain. Although there are alternative methods, such as the multi-scale modelling method [3] we have previously proposed, for fire modelling engineers still have to shorten the length of the tunnel. We refer to this as the truncation of a tunnel because engineers cannot model the entire length of a tunnel that span a kilometre. Case in point, a 1 km tunnel, with 40 m^2 cross section, using a 0.1 m grid would require in excess of 20 million cells. The computational time required to establish the fire and evacuation time line would be several weeks, which is precious design time lost.

The intent of this and the subsequent subsections is to set out the key areas to be considered when modelling a fire in a long tunnel. It is not the intent to repeat the general fire modelling good practices when using FDS as these good practices are included in the FDS User Guide [14].

2.3.2 Simplified Geometry

As FDS is based on a cartesian grid, i.e. a Lego block approach, this presents an interesting issue in how a tunnel with a curved cross section can be modelled. Engineers can, and often do consider three main approaches when modelling a tunnel.

Firstly, a simplified approach is to approximate the equivalent duct diameter, or hydraulic diameter of a tunnel and create a geometry using this approach. This approach is simple, but from experience it is infrequently used by engineers. The first reason for this is because a tunnel has various features, such as curvature where most tunnels have a curve, with the exception such as cut and cover tunnel, or immersed tube tunnel that can be rectangular.

Secondly, a real tunnel contains many features, such as cross passages openings which change the sectional profile of a tunnel, emergency walkway, maintenance walkway, enlarged caverns or cross over for train tracks, and these are all features captured in a CAD (computer aided design model). The simplified approach often would only work if the tunnel section of interest do not contain these features, and often it is faster to import the CAD models into FDS, than to build a simplified model slowly. From experience, this is a difference of a day (import and model clean-up) compared to a week or more (building manually).

The second approach mentioned above is to import the tunnel model into FDS via a CAD model. As noted earlier, even with importing such a model, time is needed to clean-up the model, that is to remove any unwanted asset (a technical CAD term used to denote items that are modelled), because most of the assets are not required in FDS, and often they can create unintended consequences such as blocking the cross section area of the tunnel, or resulting in an unintended opening on the tunnel wall.

The third is a hybrid approach where engineers firstly import the CAD models into FDS, and on top of this they would simplify the geometry in some areas for example by neglecting the presence of the walkway if the engineer believe it would not impact the modelling results.

Depending on the chosen approach, engineers would end up either with a simplified model (square or rectangular duct), or with a fully imported, or hybrid model whereby the tunnel's curvatures are represented as a stair stepped. Engineers doing this simplification (square or stair stepped tunnel cross section) should be aware of the impact this may have.

Based on a sample cross section as shown in figure 2.2, the differences between the two methods as shown in figure 2.3 show there is some difference in the mass flow between the stair-stepped (CAD, or hybrid approach), to the square duct (simplified approach). This is primarily due to the challenges in getting the same cross sectional areas between the two methods.

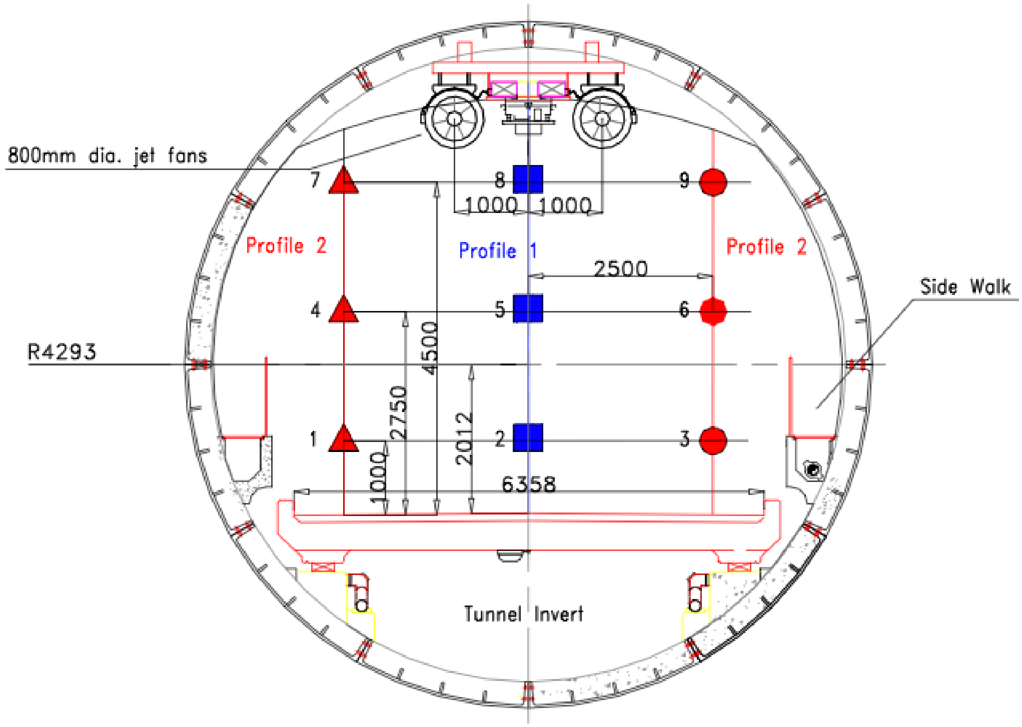


Figure 2.2: An example of a tunnel cross section [3]

Although the differences are minor when considering the actual variation in a real tunnel when the tunnel ventilation system is operated (air leakages, temperature variation etc), engineers should be aware of the possible differences especially when the tunnel cross section is smaller like those of a metro train tunnel compared to a road tunnel with a larger cross section. One method is to compare the volumetric flow rate achieved between 2D and CFD methods to ascertain the potential differences. Another method is to introduce a safety buffer in the tunnel ventilation system design.

2.3.3 Pressure Solver Settings

The configuration of the pressure solver is unique to a tunnel fire, and is the focus of Chapter 4 together with discussions in Section 2.2. Briefly, this issue is due to the low Mach number formulation of FDS which assumes instantaneous pressure propagation in a tunnel model.

As identified in our work, when using FDS for modelling fires greater than 10 MW the pressure solver will need to be configured to minimise the numerical artefacts. Numerical artefacts in

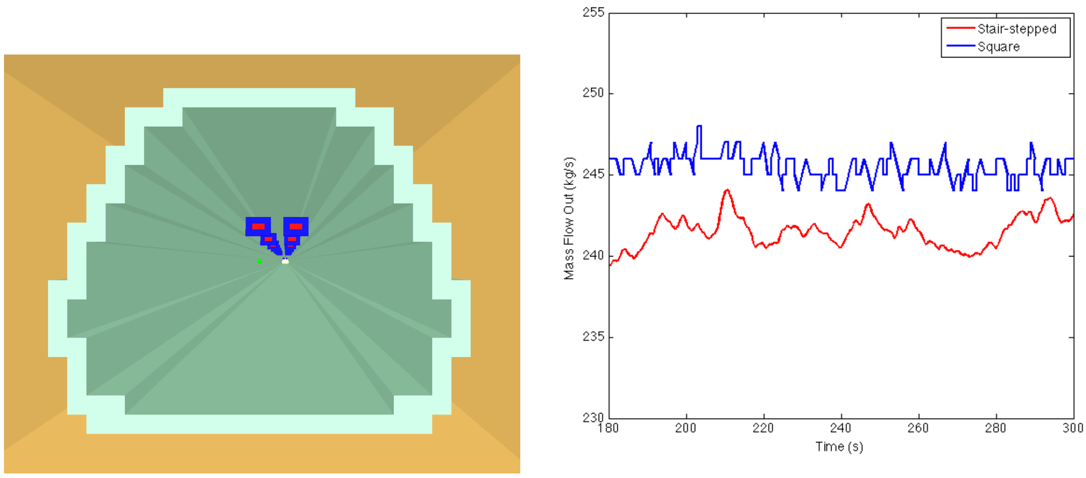


Figure 2.3: On left is the front view of the stair-stepped tunnel. On right is the comparison of the mass flow out of the square and stair-stepped tunnels [3].

this specific case mean oscillatory behaviour as shown in figure 2.4. This oscillatory behaviour is problematic for several reasons. It is misleading, when we first discovered this [16], we thought this represented a real life fire phenomena known as pulsation [24]. In addition, the oscillation shows significant variation in the mass flow rate, resulting in questions on the expected mass flow that would be used to inform the design of the tunnel ventilation system.

Upon further investigations as discussed in Section 4, we note, and this is corroborated by the FDS developers [17] that the issue lies in the pressure solver.

When carrying out simulation for tunnel fires, as a starting point engineers should consider whether a single mesh or multi-mesh is used. Although a single mesh significantly reduces the potential for numerical errors (no mesh boundary), the computational time is costly for a long tunnel, especially noting the limitation of the OpenMP [14] capability. Therefore, if multi-mesh is used, it is critical to ensure the tunnel is configure such that the newly implemented FDS solution using TUNNEL_PRECONDITIONER can be enabled [14]. This means the tunnel should only have an OPEN vent (a configuration in FDS for boundary conditions to note atmospheric condition) at each end of the tunnel, with no OPEN boundaries any where else.

Note TUNNEL_PRECONDITIONER is a new feature introduced to enable multi-mesh setup when modelling a tunnel while minimising numerical oscillation. This is discussed further in

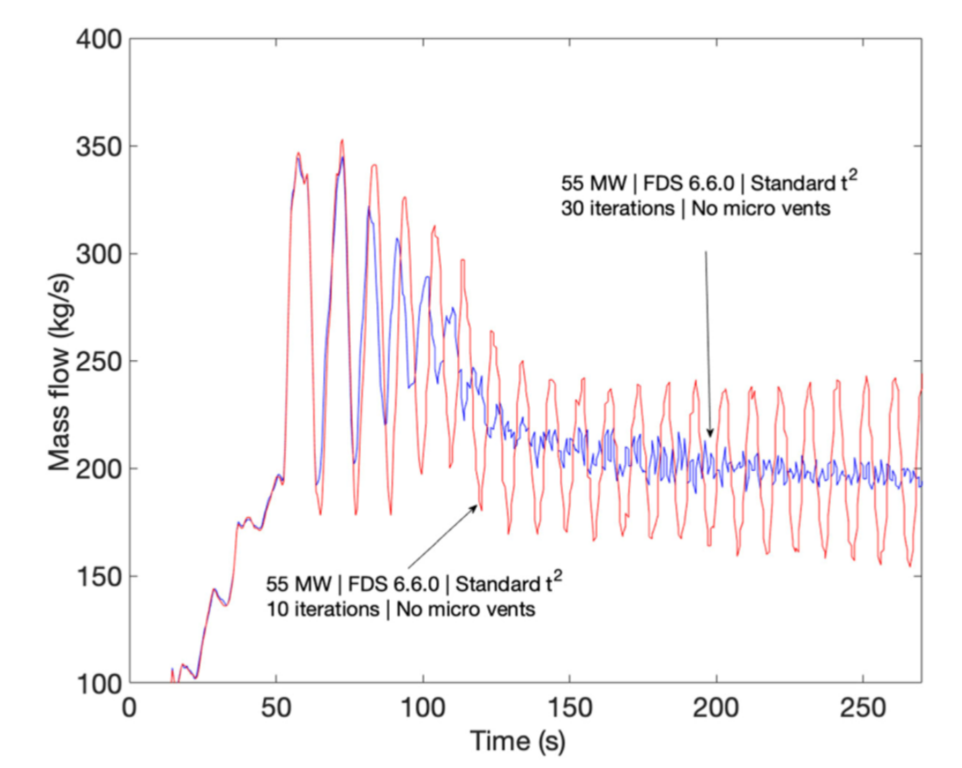


Figure 2.4: Comparison of mass flow out between 10 and 30 pressure solver iterations

Chapter 4.

On top of the above, a second item to consider is to adjust the number of pressure iterations to be greater than the default 10 (FDS increases this to 20 when TUNNEL_PRECONDITIONER is invoked). Very briefly, the pressure solver in each iteration would check for error against the previous iteration (default value to within 0.95 error), and would suspend the solver if the error is exceeded. The key here is noting the error is checked against the previous iteration meaning with the limit to 10 iterations, there is a limit to how close the pressure solver can get to the true value of the pressure. This may be adequate for a compartment fire, but as evident from the challenges in modelling a fire in a long tunnel, it is inadequate.

The authors' suggested number is 30 iterations. However, this will need to be monitored in the output file to identify whether the solver is utilising all 30 iterations consistently, or if the maximum used is under 30. If 30 iterations are consistently used, the user needs to check the velocity error within the output file. It is highly probable that a greater maximum iteration is required.

An alternative to TUNNEL_PRECONDITIONER and adjusting the maximum number of pressure iterations is to use the optional GLMAT solver [14]. From experience, this solver is computationally very expensive and is therefore unlikely suitable as a primary option for practical application due to the length of the tunnel being modelled. That said, this is an option to be considered if the pressure profile in the tunnel appears to be erroneous after applying the options suggested above.

The above options are described in detail in the FDS user guide [14]. However, the user guide includes an option of including micro-vents (OPEN boundary in one cell size) along the roof or the floor of the tunnel to alleviate pressure oscillation. This method of using one cell sized micro-vents along the tunnel is considered a numerical trick because these vents do not physically exist in the real world over an extended length in the tunnel. Therefore, this should be considered as a tool of last resort.

Even with the changes above, oscillatory behaviour is still observed, albeit with its amplitude significantly subdued as shown in figure 2.4. In engineering design, this minor variation is likely negligible as in a real tunnel other characteristics of a tunnel (surface finishes, temperature variation etc) would result in similar variations.

2.3.4 Truncation of Tunnel

As noted earlier, owing to the length of a tunnel it is necessary to model only a select section of a tunnel, instead of the entire length of the tunnel. The selection of which section to model by engineers is normally driven by where the location of a fire in the tunnel is assumed to be, the location of the cross passage exits, and the location of the ventilation shafts exhausting smoke from the tunnel.

To give an example using a metro train line, the distance between two stations is normally between 1 to 3 km. It is unrealistic to model the entire length between two stations, and often this can be simplified by the engineer selecting a section that observed the characteristics (fire location, location of exits and the location of the ventilation shafts). From experience, the modelled section is typically 300 to 600 metres, and the engineers only need to know the boundary condition at the head and tail of the simulated truncated section.

The boundary condition is easily known because a tunnel can be simplified to a simple tube with an opening at both ends. Note we have simplified the point above, and we acknowledge engineers still need to account for cross passage doors opened in an evacuation, air leakages etc (these can be represented in the model and are not discussed here to avoid complicating the point).

The boundary condition is where a mistake is commonly made. Whilst the tunnel is truncated, engineers at time neglect the fact the actual tunnel is longer than 300 to 600 m, meaning in reality when the tunnel ventilation system is activated, it takes time for air flow to be established. This is on top of allowing for the tunnel fans to be stopped, reversed and ramped up, an allowance is needed to allow for airflow to establish. Without this consideration, the model would be incorrect as in a truncated (shorter) tunnel, the air flow from a reversed and restarted fan would be established sooner than reality. This is a common mistake in engineering design where engineers often assume the airflow in the ventilation direction would be established as soon as the fans ramp up to the maximum speed.

In a transient model considering the tenability of passengers in a fire, this incorrect earlier establishment of airflow can represent an incorrect (overly optimistic) scenario.

There are a couple of methods to obtain this allowance for airflow to establish. The first method is by obtaining this information from a 1D model, e.g. SES or subway environment simulation [25], a bespoke 1D computer program used regularly in design. Alternatively, by knowing the volume of airflow supplied into the tunnel, it is possible to obtain an approximate time for airflow to be established from one end to the other. Recently as a good practice, engineers have started the use of indicating the timeline between a tunnel ventilation system operating in reality, and the modelling time line. See figure 2.5.

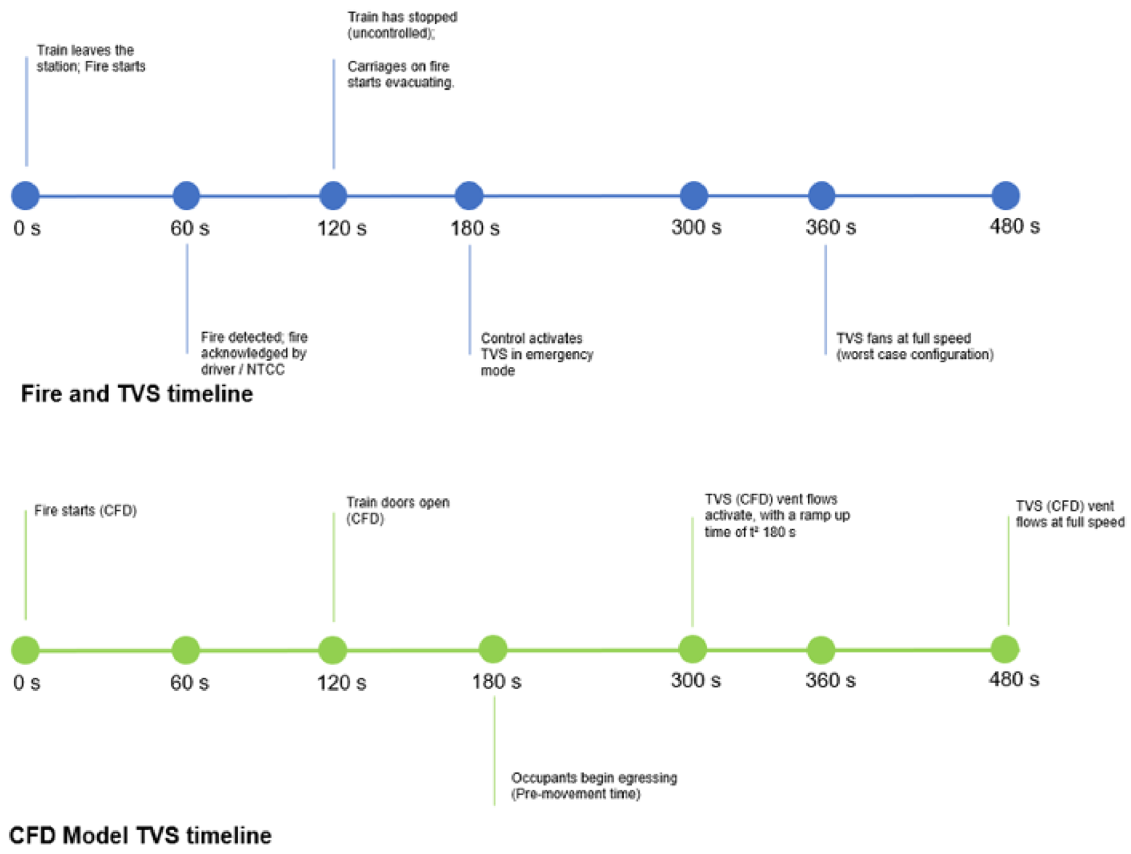


Figure 2.5: Comparison of timeline used in a CFD model to an operational timeline [4]. Top timeline represents the sequence of operation for a tunnel ventilation system (TVS) in a fire, and bottom represents the timeline in the CFD model.

2.4 General FDS Settings and Validation

This section contains the general inputs when FDS is used throughout this thesis. Any modifications, or specific inputs will be discussed separately in the relevant chapters.

FDS versions 6.5.3 to 6.7.6 have been used throughout this thesis. The thesis research started in 2016, and the FDS version changes reflect most recent versions available throughout the research period.

The LES, or large eddy simulation solver is used for all the models in this thesis. LES is the primary solver in FDS and is the main solver considered in the FDS validation guide [14].

A methane based fire, in an ambient environment temperature of 20°C are used for all the fire models in this research except for Chapter 3 investigating the carbon monoxide modelling. For the choice of methane fire in the other chapters, as the primary objective is heat generation, a simple methane fire is selected to reduce the variability associated with using a comparatively more complicated material, such as timber or polyurethane.

A key consideration for FDS and the unique properties of LES are the consideration on whether the correct mesh has been selected. Note correct in this instance means whether the mesh selected is adequate to represent the expected physical reality of a fire. Whilst FDS has published a comprehensive validation guide [15], the guide primarily contains compartment fire and small fires only. Therefore, where practicable, we have compared the CFD model to the validation case, this is relevant for Chapter 3. In Chapters 4 and 5, we have considered the grid independence of the mesh size.

The reason being in the FDS validation guide, there are only a handful of cases for tunnel fires, and in all instances the fire size considered is smaller than those we have modelled in these chapters. Therefore in these case, we have considered the grid sensitivity analysis, and in the case of Chapter 5, comparing to another CFD solver.

Through this thesis, we have reported the simulations with different versions of FDS. We have re-run the models using a newer version of FDS where we believe there is a material difference, e.g. in modelling tunnel fires where the newer version of FDS reduces the numerical oscillation. However, in other cases, e.g. assessing the CO production, we only presented results carried out with the original version used as we do not believe there has been significant changes that would change the conclusion of our work.

This issue occurs in engineering design where it is impractical to always re-run the CFD models using the latest versions of the CFD package. This is particular the case for projects that span several months where a new version could be released in between. It is important for engineers to understand and decide if the new version contains fixes of errors that could skew the result. Engineers should not be expected to re-run all models without considering this, nor should they blindly keep to the original version without understanding the changes.

2.5 Conclusions

In this chapter, we discussed the relevant characteristics of FDS when modelling tunnel fires. The primary reason increasing simulation complexity is due to the significant length of the tunnel when compared to a compartment fire. This significant length, coupled with the curvature expected in a tunnel (noting FDS is based on a cartesian grid), presents unique scenarios that need to be considered when modelling a tunnel fire.

We showed in this chapter the key issues to consider. Firstly it is to consider the method to represent the geometry of the tunnel which suits the need of the engineer. There are cases, as shown in Chapter 5 for example where a duct representation is adequate.

Secondly, it is critical to consider the settings of the pressure solver, and the set up of the meshing, whether it is a single mesh or multiple meshes when modelling a tunnel fire. Due to the extended length of a tunnel, and the incompressible flow configuration (pressure propagates instantaneously where in reality they travel below the speed of sound), the pressure solver settings when modelling a fire in a tunnel needs special attention. Otherwise, this results in unexpected pressure oscillation, or numerical instability, both of which render the results useless, or dangerous as it could misinform the engineer.

Thirdly, we have illustrated the common engineering mistakes made when only a truncated tunnel section is simulated due the limitations of computational power and time. When truncating a tunnel in a model, engineers cannot assume that the flow in a tunnel would be established instantaneously in the truncated section, as flow would still need to travel from one end of the tunnel to another in a longitudinally ventilated tunnel. We suggested the use of a timeline comparing the actual operation sequence of a tunnel ventilation system in a fire, compared to the timeline used in carrying out the CFD model.

Lastly, we covered the general settings of FDS used in the modelling carried out throughout this thesis. A key issue we raised is the need for validation due to the unique nature of LES, compared to other turbulent models such as RANS.

Chapter 3

Carbon Monoxide Modelling

3.1 Understanding Limitations of CO Modelling

CO, or carbon monoxide is known as one of the main causes for fatalities in fire incidents. Because of this, and with the advancement in CFD modelling over the last decade, CO is used as a design variable in practical application to assess the tenability for occupants in a fire scenario.

This chapter includes a brief introduction on the role of CO and its consideration in practical application, including its modelling. We show here the challenges associated with modelling CO due to its inherent large variability in experiments.

Some of the preliminary work in this chapter was presented at the Asia-Oceania Symposium for Fire Science and Technology [19].

3.2 Background to Our Work

In the practical application of fire safety engineering design, fire modelling is often required to understand the impact smoke has on the evacuation and firefighting operation. In this analysis, a concept commonly used is the tenability criteria based on the results from fire modelling. Engineers in Europe, Asia Pacific and the United States often consider numerous variables including smoke temperature, visibility through smoke, radiant heat emitted by the fire, and CO (Carbon Monoxide) toxicity, is the focus here.

With toxicity, the main component used in the design is CO. This is because CO [5] has been responsible for over half of all fire fatalities and represents the most common fire toxicant.

Fire Dynamics Simulator (FDS) [14] is a popular computational fluid dynamics (CFD) package for fire and smoke modelling. The user can dictate the rate of production of CO via the CO yield variable.

CO yield is a key dimensionless parameter used in fire safety design [5] to represent the proportion or percentage of a material that converts into CO in the combustion process. This flexibility in control is important when designing for various fire scenarios. FDS, like other solvers is based on numerically solving turbulent flows via approximations. This is an important point because a computer model is essentially a simplified representation of a physical model. Even with refined mathematical solution techniques, compromise and simplification are needed as reflected by researches over the last decade on numerical prediction of combustible products [26, 27].

The CO yield used in FDS modelling is not a parameter that can be directly observed or measured in an experiment [5]. Beyler [28] is noted as the first researcher to publish major species production rates in an experiment using a burner with a collection hood. The CO yield is derived from measurements in experiments, for example FTIR measurements, with the resulting measurements correlated to a global equivalence ratio (GER). The GER represents the overall ventilation of a control volume from an experiment, and the CO yield being derived as a ratio from the GER. Although this process is in a controlled environment, the combustion process itself introduces variability to the measurements.

This is the premise of this research. As shown by Melcher et al [7] there is a large variance in the CO concentration and its measurement, and the challenges associated with repeating the experiments to achieve comparable CO yield.

We demonstrate via a priori model using FDS6.6 the complexities in modeling the CO generation where we compared our results to the CO measurements by Melcher et al [7]. We show here the variability in predicting the CO generation in different phases of a fire, from inception, growth to decaying, and the limitations in predicting the CO concentration during the decaying phases of the fire. This is not a critique of FDS, but instead to highlight challenges in numerically predicting CO.

Following the a priori model, we showed with posteriori models by firstly calibrating the FDS model's combustion (mass loss rate) and CO production to one of the two known Melcher *et al* [7] experiments, i.e. Series S2. With the calibrated model, we then use this to run the second of the two known Melcher *et al* [7] experiments, i.e. Series S3 where all the materials and configurations remain the same except for an increased in the materials burnt. The results are presented and discussed.

3.3 The Criticality of CO in Fire Safety Design

CO is commonly used in practical application when assessing the tenability of occupants in fire safety design. CO is often present in a real fire due to the incomplete combustion, and when occupants inhale CO during evacuation in a fire, CO mixes with the oxygen in the bloodstream to create carboxyhemogoblin, which is a stable compound. This compound then prevents the transport of oxygen in the bloodstream, and excessive exposure reduces the oxygen carrying capacity of the bloodstream thereby resulting in incapacitation due to oxygen deprivation. This fatal aspect of CO is the first factor driving its consideration in practical application.

Secondly, whilst it has been possible to estimate the presence of a fire in combustion using manual (hand) calculations [5], this is rarely done because CO as a parameter is only of use when considered from a transient perspective over an egress path. This is because CO inhalation and its lethality increase over time, and a temporarily short period of exposure would not result in an injury or incapacitation. A non-transient model cannot take into account this short period. Therefore, the adoption of CO as a design parameter started taking off when this parameter

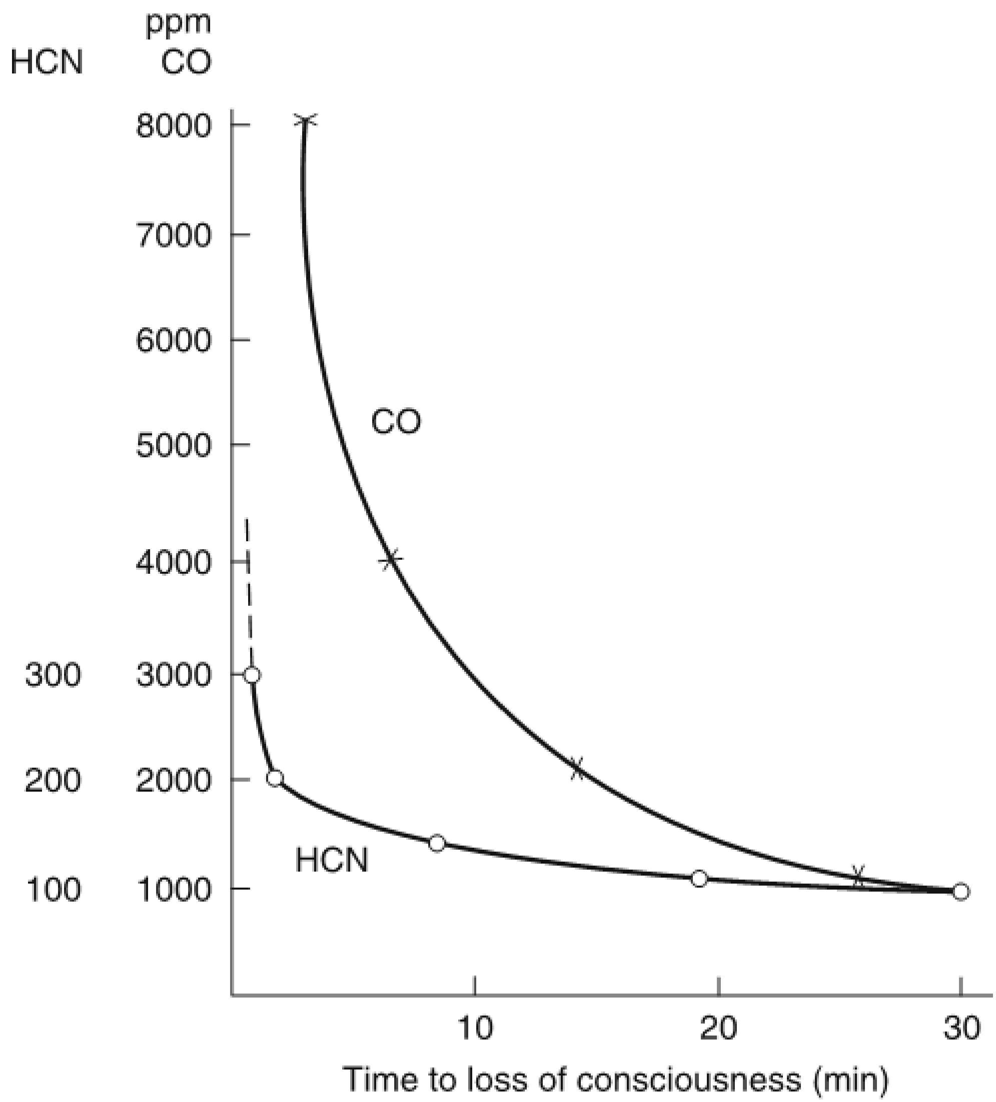


Figure 3.1: Exposure to CO and the time to loss of consciousness [5].

and its transient production could be modelled in CFD such as using FDS.

From a practical application perspective, CO is assessed as a transient variable, whereby its concentration over an egress route is modelled over time. Designers then compare the exposure of occupants to CO during the evacuation period, and assess whether the condition is tenable. The tenability is often benchmarked to an acceptance (pass or fail) criteria [5]. An often used benchmark is exposure to 100 PPM (parts per million). An example is shown figure 3.1.

The pass or fail criteria used above, whilst simplistic do not recognise the inherent variability to loss of consciousness as the time would vary depending on the health and age of those exposed.

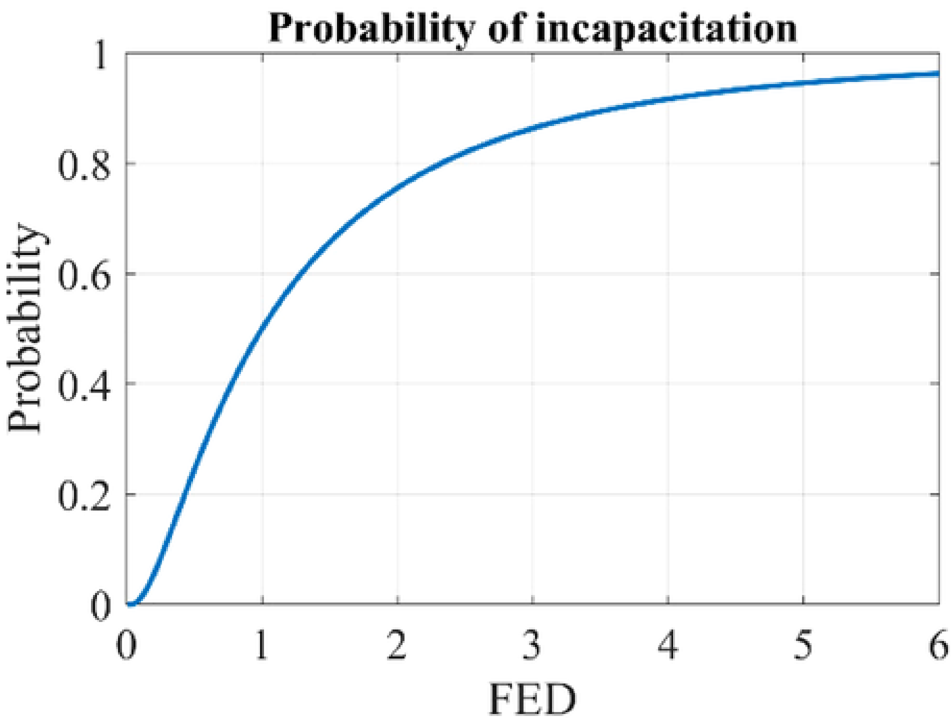


Figure 3.2: ISO 13571 curve showing the probability of incapacitation [6].

Because of this, the alternative approach, the FED, or fractional effective dose approach is used whereby the exposure of occupants to CO is measured as cumulative exposure over time and across the egress route, and the exposure level is compared to a tenability scale whereby prolonged exposure increases the likelihood of an occupant being incapacitated. A typical FED factor used is 0.3, whereby it is simply a ratio of CO dose received over time divided by the dose required to cause incapacitation. This value represents the likelihood of incapacitation, a probabilistic consideration, not absolute and a value closer to 1 represents greater likelihood of incapacitation.

The FED is simply a ratio, and the calculated FED is benchmarked against standards, such as ISO 13571 [29] that suggests an FED of 0.3 has a probability of 11% of healthy adults being incapacitated shown in figure 3.2

At this point, it is clear there are two major assumptions here. Firstly, due to legal and ethical reasons, there has been limited research on the health effect when a human is exposed to various level of CO. Historically [5], majority of these researches are based on exposure to lab mouse or derived statistically as an estimate from past incidents. This means the FED level is at best a guesstimate of CO tenability with a large range of uncertainty.

Secondly, the estimation of CO as demonstrated in the experiments by Melcher *et al* [7] shows there is a large variability to the CO production when the experiments are repeated.

The subsequent sections shows we need to understand the physical phenomenon of what we are modelling, and these physical uncertainties would carry into and be amplified when modelled.

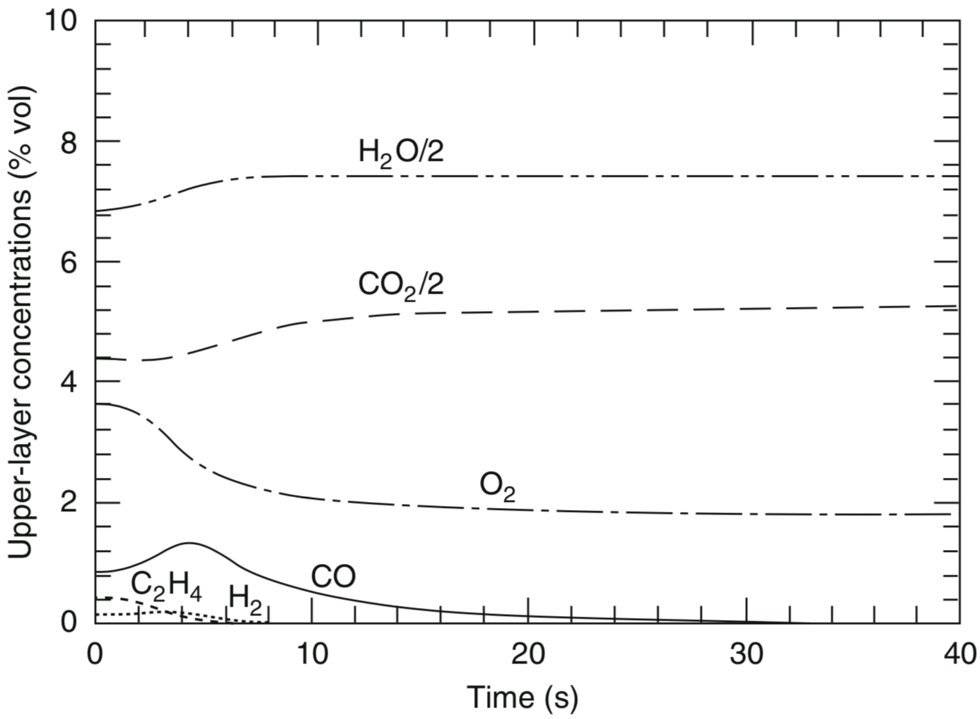
3.4 Formation of CO in Combustion

The phenomenon of species product formation, specifically on CO production in real fires is comprehensively documented by Gottuk and Lattimer [5]. This is summarised here briefly to provide contexts to the research.

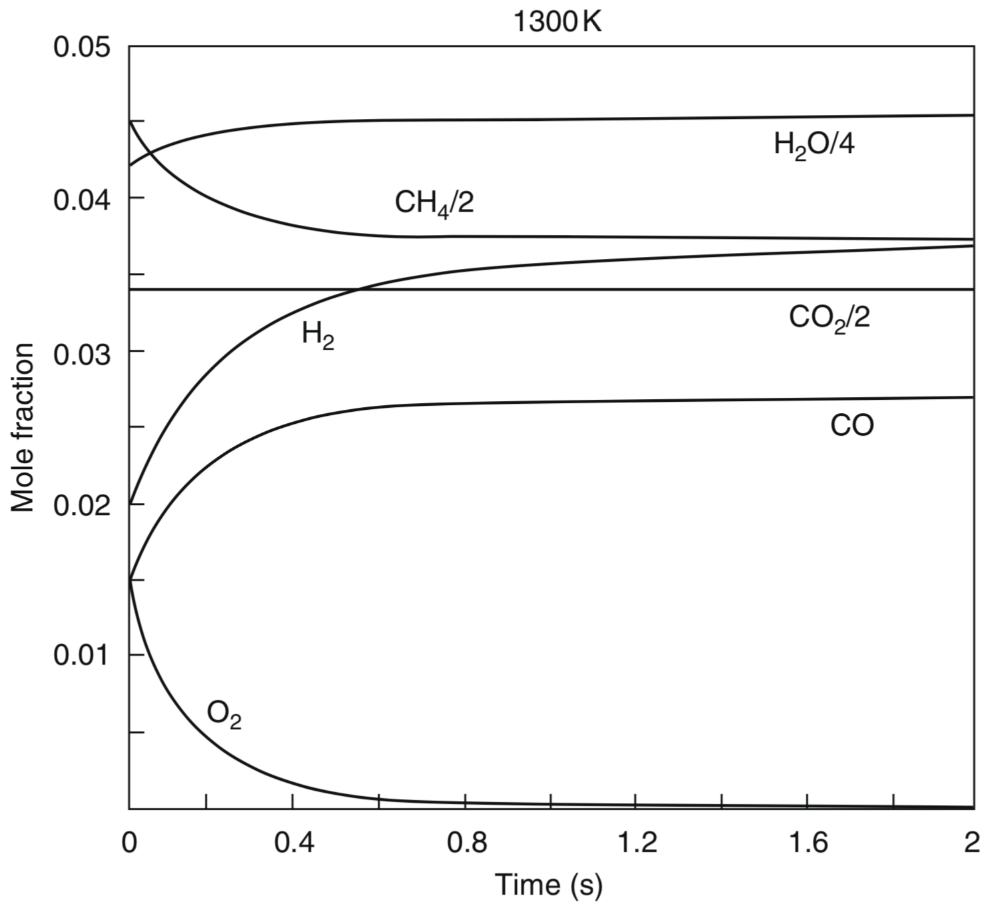
Simplistically, for common hydrocarbon-based fuels, in an oxygen rich environment with perfect combustion, negligible formation of CO is expected and CO_2 and H_2O are the main products. In a real fire, e.g. a fire in a room or in a confined space such as a tunnel, as the fire grows it transitions into a ventilation-controlled fire and CO is produced. In a ventilation-controlled fire [1], there is a large formation of CO. As the oxidation of hydrocarbon (fuel) occurs before CO (by product of combustion), in a ventilation-controlled fire with limited air (oxygen), the oxygen level is depleted before all the hydrocarbon is oxidised leaving CO in the smoke plume, rather than fuel and CO being oxidised completely in a perfect combustion where there is abundant oxygen, i.e. not a ventilation controlled combustion.

3.4.1 Experiments by Melcher

Briefly, Melcher *et al* [7] investigated the repeatability of real scale fire tests with three series of experiments with one involving n-hexane as a fuel source, and the remaining Series 2 and 3 (S2 and S3) using spruce and fir woods. Series 2 and 3 are the experiments of interest in this chapter.



(a)



(b)

Figure 3.3: CO concentration [5] a) in an oxygen rich environment and b) in a ventilation controlled environment.

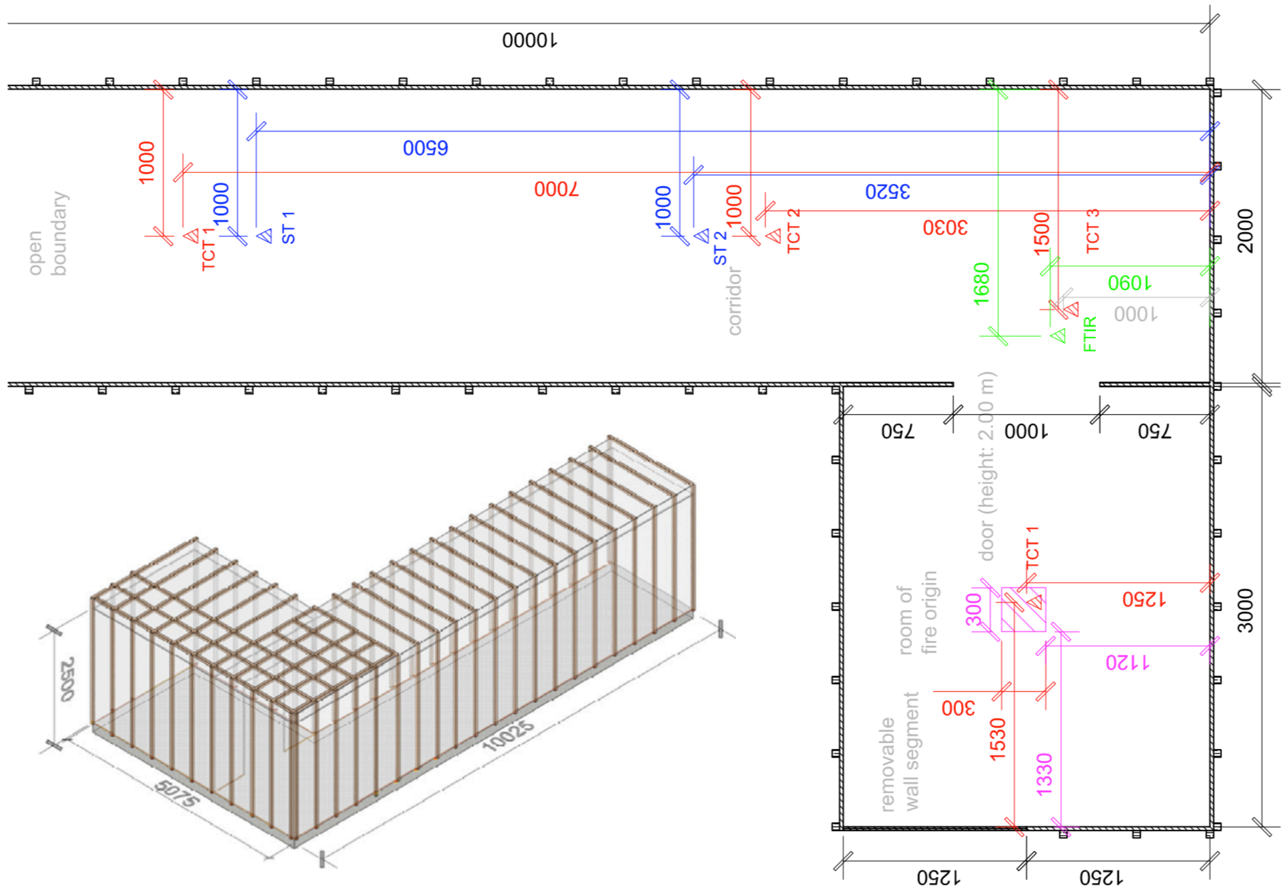


Figure 3.4: Real scale experiment rig by Melcher *et al* [7]. Note all measurements are in millimetres.

For the wooden fuel in Series 2 and 3 experiments, eight scantlings (24 by 48 by 300 mm) were assembled into a crib. Three of the cribs were used in Series 2, and 5 for Series 3 experiments. To test the repeatability, the Series 2 and 3 experiments were repeated twenty times, each test lasted 30 minutes and the rig was recalibrated before next. The experimental rig is shown in figure 3.4.

Following the twenty repeated tests for each series, Melcher *et al* [7] showed the range of CO concentration measurements, together with the mean and median in figure 3.5. The CO concentration for the repeated experiments are measured outside the room where the crib is located. See figure 3.4. The measurement is done using FTIR, or Fourier Transform Infrared Spectroscopy, where FTIR is a suitable method to measure infrared active gases such as carbon monoxide.

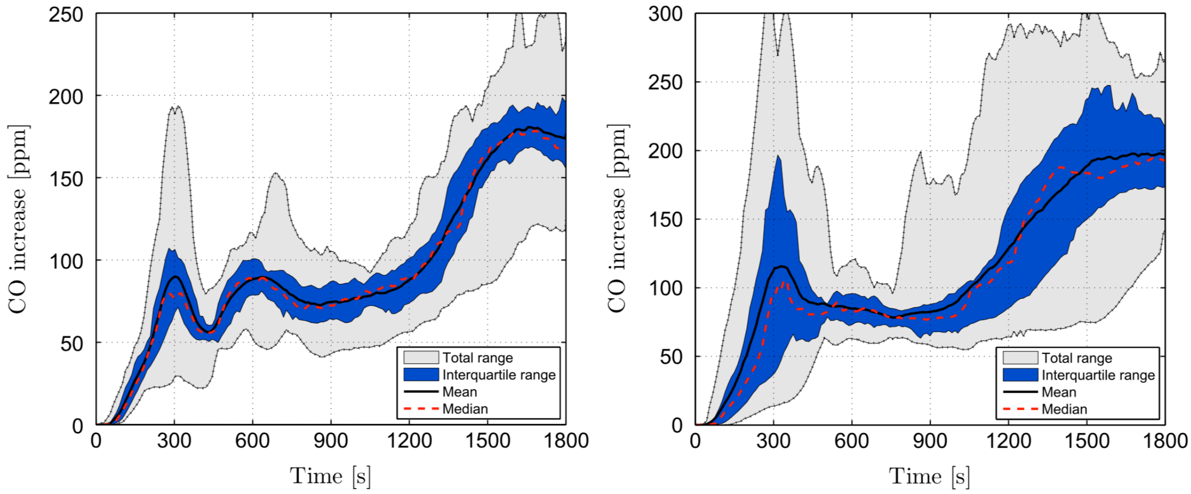


Figure 3.5: CO measurements in Series 2 (left) and Series 3 (Right) experiments by Melcher *et al* [7].

As each gas has a specific infrared spectrum, the measurement in the form of signal pattern is then compared to a reference pattern (air sample) to determine the concentration of various gases. Although not explicitly explained by Melcher, this difference in comparison between measured sample versus air sample is likely the reason why the phrase CO increase is used in the y axis as seen in figure 3.5, as this is a comparison of elevated CO concentration (increase) compared to the air sample. The CO concentration in the y axis of figure 3.5 is therefore absolutely measurements at each point in time, rather than cumulative, as the graphs would otherwise show only ever increasing concentration.

3.4.2 Numerical Modelling

For our numerical model, we used FDS Version 6.7 [14]. We built the model geometry, including the FTIR and sensors' locations as per the experimental rig. A single 0.025 m mesh (dx) was used with approximately 5 million cells. The characteristic fire diameter (D^*) for the smaller S2 fire is 0.562, resulting in a $\frac{D^2}{dx}$ ratio of 22.5. Benchmarked against FDS Validation Guide [15], the chosen mesh is within the numerical parameters of the NIST FSE experiment. This provides confidence the selected mesh parameters are within the non-dimensionalised parameters derived from an experiment.

With the experimental rig now modelled in FDS, we considered there are four variables influencing the production of the CO in the context of the Melcher *et al* [7] experiments. These variables are the material type, the combustion, the CO yield specified and the heat of combustion.

Firstly the material type or chemical composition could influence the CO generated. For the FDS model, we have specified the material composition as provided by Melcher *et al* [7]. This variable is therefore controlled.

To replicate the configuration of the wooden cribs, the total exposed surface area of the crib is estimated at approximately 0.21 m^2 for experiment S2, and replicated as a block in FDS, with all but the surface Z in contact with the floor being modelled as a combustible surface. This surface area is increased to 0.31 m^2 for the S3 experiment.

For combustion, given there are two sets of experiments (S2 and S3) by Melcher *et al* [7] providing experimental data on mass loss rates, we have calibrated the mass loss rate in FDS to experiment S2. See figure 3.4.2. This variable is therefore controlled. We have carried out several simulations for the calibration. Although for this calibration the mass loss is at the higher than median mass loss compared to the experiment, we proceeded with this noting that the resulting CO measurement would be biased to be higher due to the greater rate of combustion.

On the heat of combustion, this parameter is noted as approximately 18.2 kJ/kg by Melcher *et al* [7]. However, physically for the pyrolysis of wood this value is influenced by the moisture content of the material, and a range of values between 16.6 to 19.5 kJ/kg is noted in publications [5]. Therefore, we have carried out a sensitivity study for this value considering the lower end and higher end, together with the value of 18.2 kJ/kg. As shown in figure 3.7, the simulations show that the heat of combustion is not an influencing variable in CO production. This variable is therefore controlled.

Given that FDS is based on LES (large eddy simulation) where turbulent eddies are simulated, we repeated an entirely same simulation three times to demonstrate that the issue with

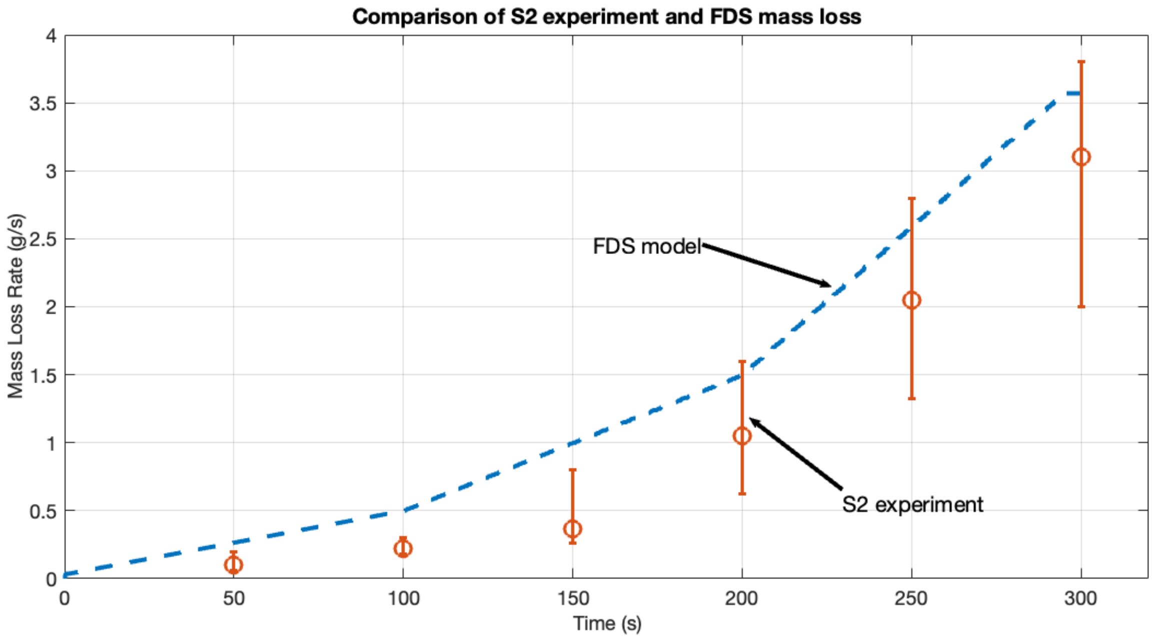


Figure 3.6: {Comparison of S2 experiment and FDS mass loss. Note the S2 experiments show the median and interquartile range.

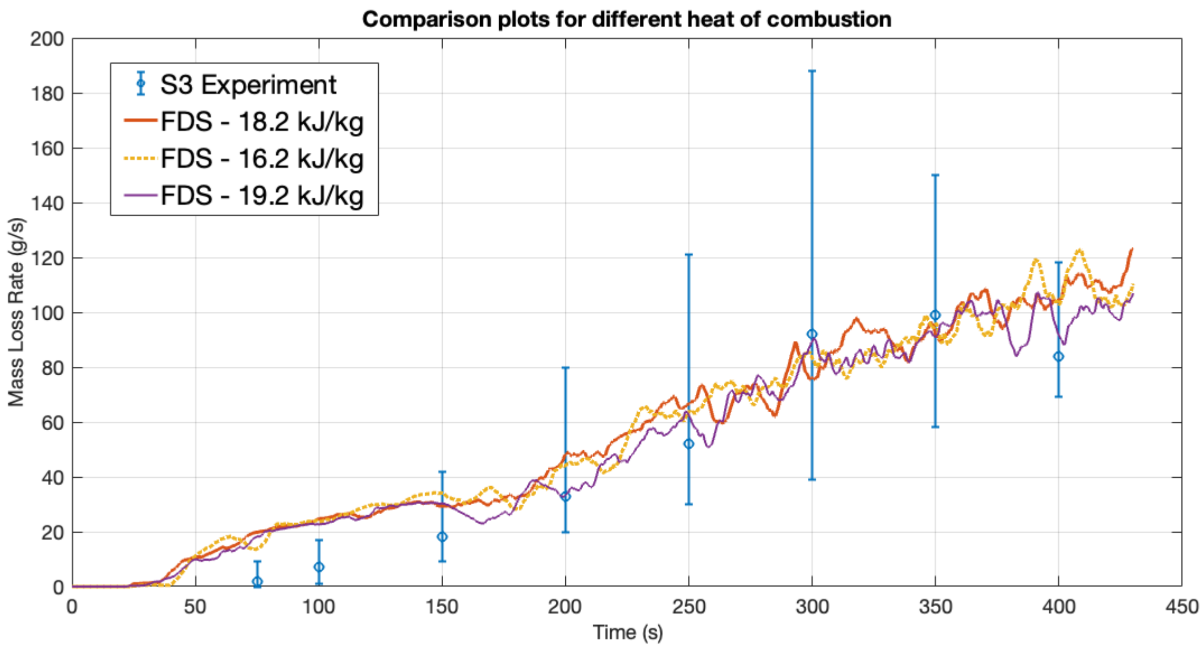


Figure 3.7: Comparison plots for CO concentrations with three different heat of combustion variables. Note the S2 experiments show the median and interquartile range.

Figure 3.8: CO measurements in three separate FDS simulations with the same parameters.

repeatability does not occur with FDS simulation. This is shown in figure 3.8.

With the above variables controlled, this brings in the last variable, i.e. CO yield, that we consider would influence CO production. The CO yield in FDS is the ratio of mass of CO in a combusted product, divided with the mass of fuel that has combusted [22]. CO yield in experiments is reported in the same manner in various publications such as the SFPE Handbook [5] that is used in practical application, or by other researchers [30].

For wood such as spruce, fig or douglas that are used by Melcher *et al* [7], the reported CO yield ranges from 0.005 to 0.021 for oxygen rich combustion, as opposed to ventilation controlled combustion whereby the CO yield is at least an order of magnitude higher resulting in denser CO concentration. The higher values represent the combustion transitioning towards, but not yet into, ventilation controlled. SFPE Handbook [5], which is a widely used handbook in fire safety design provided a value of 0.005 only, and the range of values from 0.005 to 0.021 are obtained from other published researches [31] [30].

Hansen and Hull [31] noted that charring behaviour in wood also results in a higher CO yield, and charring occurs over time through the combustion process. This is noted by Bartlett *et al* [32] where the duration of combustion influences the combustion behaviour of wood due to the

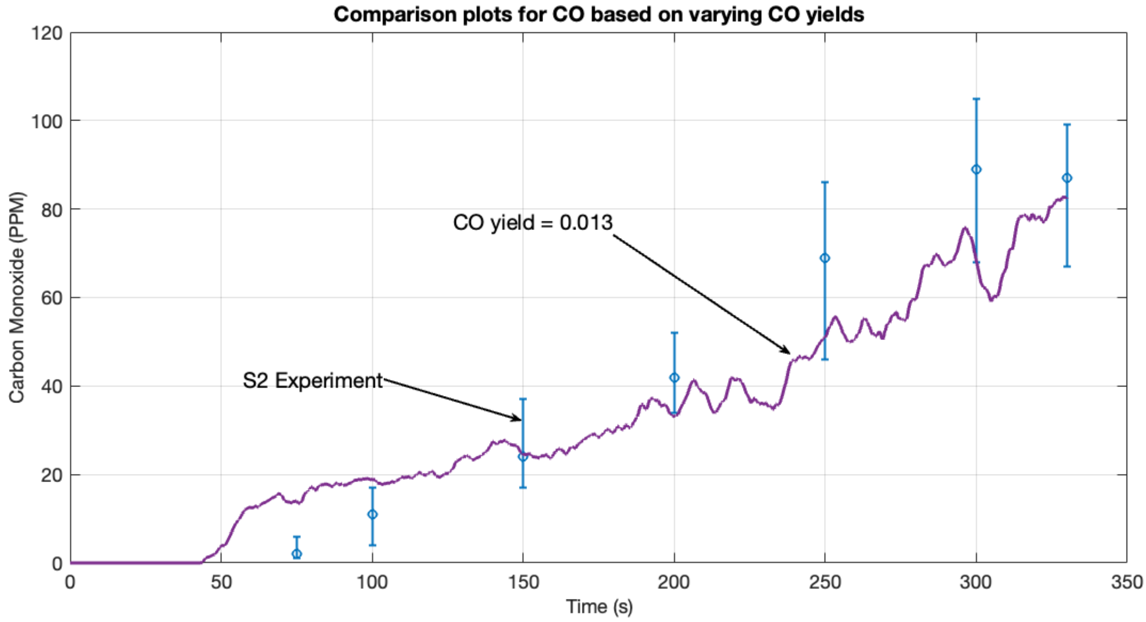


Figure 3.9: FDS CO measurements based on CO yield of 0.013 compared to S2 experiments. Note the S2 experiments show the median and interquartile range.

gradual release of of moisture from the wood.

To reduce the uncertainties due to time, and the changed combustion behaviour due to the occurrence of flashover in the experiments, we have reduced the simulation time to just before flashover as noted by Melcher *et al* [7]. This is 330 seconds for S2 and 420 seconds for S3. By comparison, the experiments' duration is 1,800 seconds. A shorter simulation time prior to flashover will reduce the uncertainties associated with the change in pyrolysis and the occurrence of charring that would introduce uncertainties in the production of CO.

Having tested a range of CO yields between 0.005 to 0.021, we showed in figure 3.9 that a CO yield of 0.013 produces CO results reasonably close to the overall S2 experiments.

3.4.3 Simulation Results

Using a CO yield of 0.013, this has been used to simulate CO production in S3 experiments. The same mass loss rate, but with an increased surface area to account for the increase in wood cribs is used. A heat of combustion of 18.2 kJ/kg is used. In addition, three other CO yield

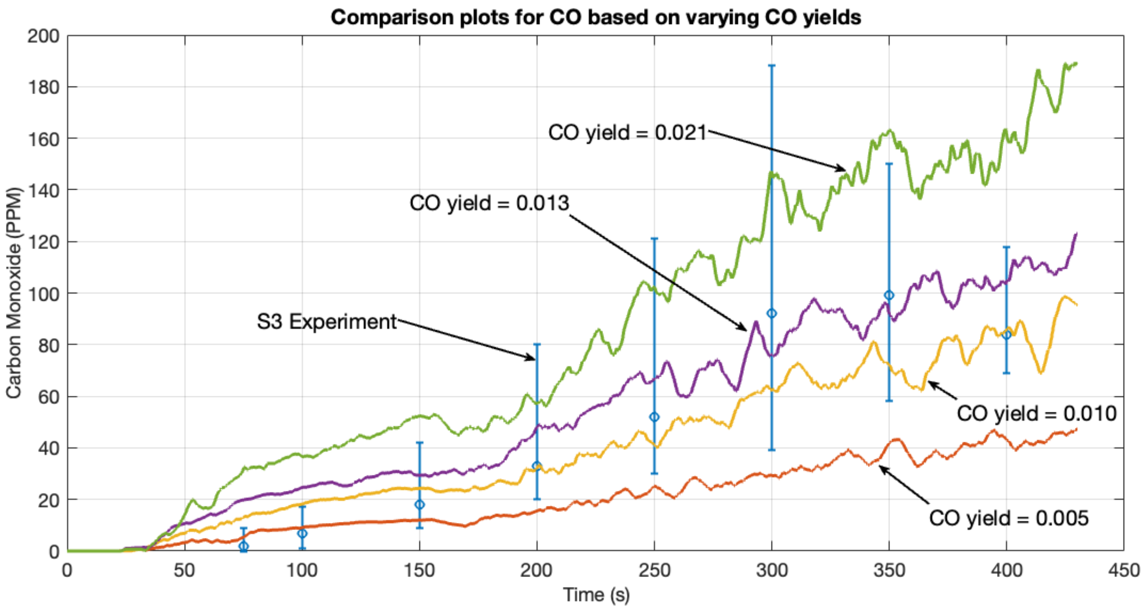


Figure 3.10: Comparison of FDS simulations with S3 experiments. Note the S3 experiments show the median and interquartile range.

has been used in the simulation to provide a benchmark. These are 0.005, 0.010 and 0.021, on top of 0.013. See figure 3.10 for the results for S3 experiments. See figure 3.11 where a similar comparison simulation has been produced for S2 experiments.

For the S3 simulations, the mass loss rate based on the calibration in S2 has been used as noted earlier where we have only increased the area of the surface being burnt to account for the additional wood cribs in the experiment. See figure 3.12 that shows the mass loss rate that has been calibrated to S2 achieving a fair fit with the measurements with S3. Although this is already known in good practice for CFD simulations, this reinforces the purpose and benefit of calibration in a simulation.

In the figures showing the comparison to the experimental measurements, we have opted to use the interquartile range and the median, as opposed to the full range of the experimental measurements by Melcher *et al* [7]. This is because as shown in figure 3.5, the variability in the range is significant whereby the highest and lower CO values differ by a factor of 3, or more in some instances, rendering the purpose of a comparison to the simulation results moot as all simulations would likely fall within this range. This variability in the range of experimental results show the challenges associated with deriving a value of CO yield that could be used for

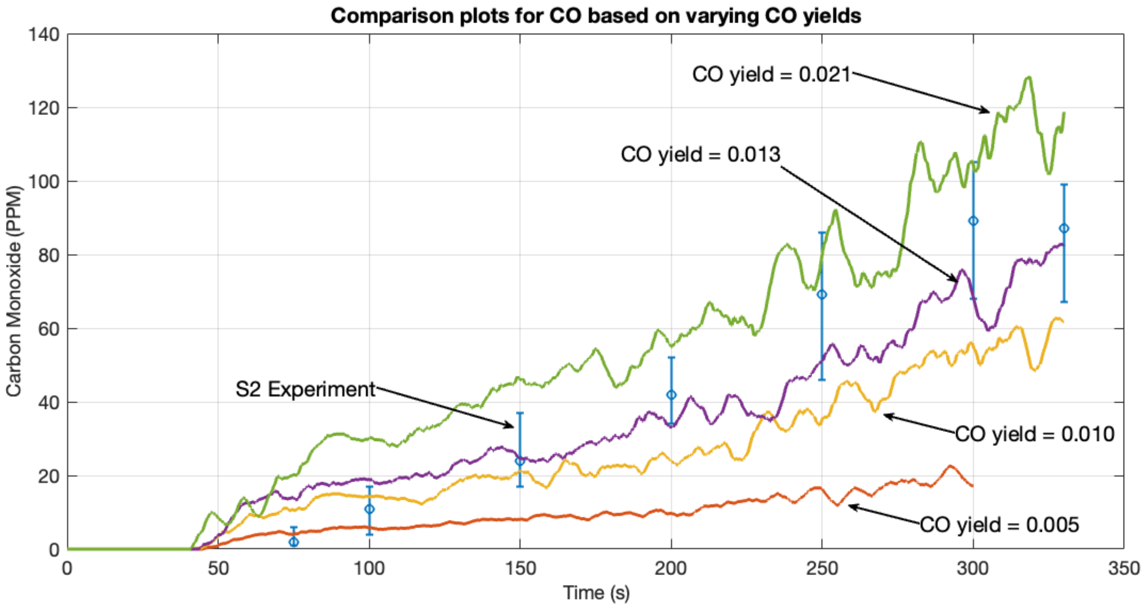


Figure 3.11: Comparison of FDS simulations with S2 experiments. Note the S2 experiments show the median and interquartile range.

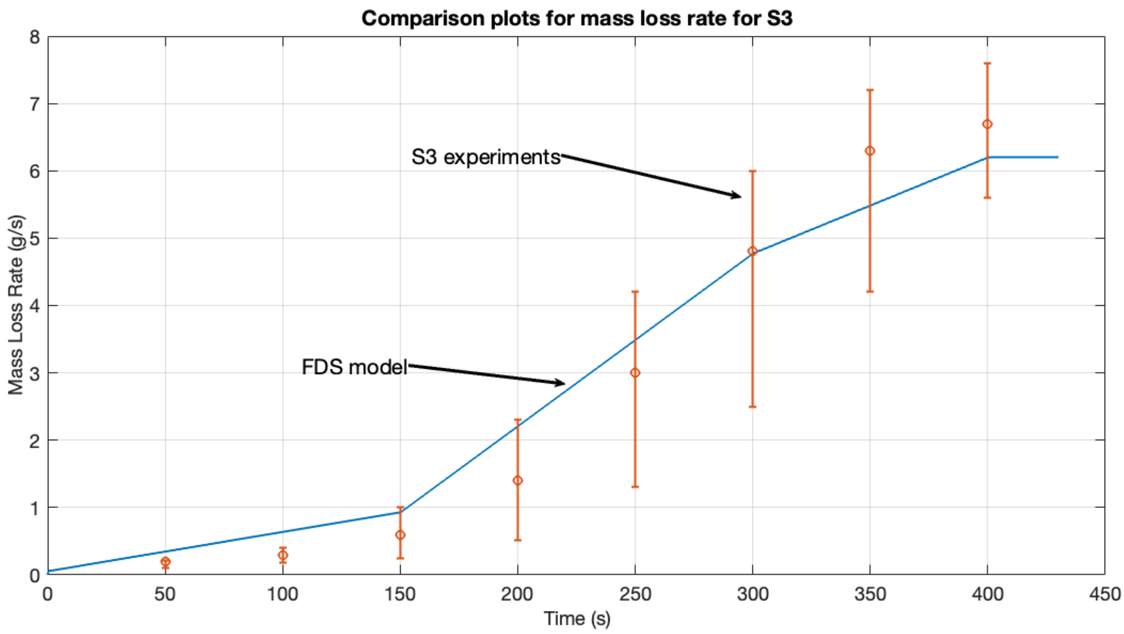


Figure 3.12: Comparison of mass loss in FDS simulation to S3 experiments. Note the S3 experiments show the median and interquartile range.

simulations.

3.4.4 The Impact on Practical Application

With the results above, we have made several observations. Firstly, with FDS, we showed that once the material is controlled, the greatest influence on CO production is the CO yield.

The CO yield of 0.005 as recommended in SFPE [5] for wood has been shown to result in an under-prediction of CO production in both the S2 and S3 experiments. The calibrated CO yield value of 0.013 results in a fairer fit to the experimental measurements. Conversely, with a CO yield value of 0.021 which is noted in researches [30], the results show an over-prediction of the CO production in S2 and S3 experiments. This shows if an engineer were to use only the SFPE value of 0.005 [5], the simulations would have under-predicted the CO production, and this could have significant impact in a design if the engineer were not aware of this variability.

These results demonstrated the challenges in the modelling of CO production in a simulation as the range of CO yields published can result in such a significant variability in the CO production. This demonstrates the challenges in situations where a range of CO yield is published for a material type, and there is no separate experimental models that can be used to select a suitable CO yield.

The unpredictability of CO modelling in numerical simulations is compounded because in practical application, CO is used in a time dependent analysis as discussed earlier, e.g. Available Safe Egress Time benchmarked to Required Safe Egress Time.

The unpredictability of CO yield across the fire growth timeline means engineers are unable to accurately verify if the CO yield predicted in modelling is reliable, or appropriate for design. This issue is further complicated by CO concentration being linked to exposure using the fractional effective dose (FED) method, whereby the tenability criteria of 0.3 maximum often used is a complex benchmark on its own, not withstanding the limitations of the numerical simulations.

This means the unpredictability of CO, coupled with the limitation of FED resulted in uncertainties being stacked on top of uncertainties. In practical application, a design needs to be safe so far as is reasonably practicable. Engineers will need to balance overdesign and achieving a so far as is reasonably practicable standard of safety, as otherwise designs will be too costly to get off the ground. This challenge means in practical application it is not simply the case of using an onerous (higher) value, also noting the earlier point on choosing a suitable CO yield, i.e. which is the best fit value.

Additionally, a further concern is that wood is a regularly studied material in experiments, and Melcher *et al* [7] showed the fire community that repeatable experiments involving wood remain a challenge. This is not a flaw of the experiments, but is a recognition that experiments involving measuring concentration are dependent on many variables, from material composition which is influenced by the environment it is produced or grown, to environmental factors such as humidity. In the preparation of this chapter, we have also noted that whilst there are publications [32] [33] [34] on the time-temperature and fire resistance properties of timber, but there remains limited recent papers on CO yield of various materials, including timber which is one of the more commonly studied material in fire safety science.

As such, numerical fire simulations founded on experiments of wood combustion are also suspect; but only for certain species values such as CO. This is because the fire science community has fair experience with wood combustion prediction from a temperature perspective as noted earlier. In addition, even for a common material like wood the gas species production such as CO yield remains a complicated topic. This is because the wood in question for this research is untreated timber. Hansen and Hull [31] has showed that the presence of fire retardants treatment, that is the treatment to the surface of the wood to improve its fire safety related properties, can result in materially different CO production. The researches by Hansen and Hull [31] showed FRP, or fire retardant pressure impregnated wood produced twice the CO compared to untreated timber.

This means the prediction of CO using other fuels is likely to be more complicated owing to a lack of data. This is especially true for the synthetic materials found in furniture and storage in buildings and train carriages operating in tunnels. The complexities involved in numerical modelling of practical application problem are known, as demonstrated in [35, 36].

Therefore, we note in practical application, numerically predicted CO concentrations for practical applications should only be used as a secondary indicator. We propose the principal tenability criteria used in practical application should be based on parameters we have a higher confidence in predicting, such as temperature [15].

That said, as engineers, we recognise the need to appraise the various variables that could impact occupants in a fire as this provides a more complete picture of the situation. This is why we noted CO could be used as secondary indicator.

As shown in the simulation results, when we tested a range of CO yields published in the literature [5] [30], we need to consider the sensitivity of the CO production to our results. When CO production is used as a variable in design, a sensitivity study should be carried out to assess the impact the variability may have on the simulation results. Our recommendation is to consider the lower end, median and the higher end value of the CO yields as we have used here.

If a range of CO yield is not available, we recommend consider the use of a safety factor. It is the engineers' judgement to determine a suitable safety factor balancing the need for practicality and safety. Anecdotally and limited to the simulation of wood, a factor of safety of 2 should be a fair starting point. Separately, when considering the choice of CO yield for practical application, the CO yield closer to a ventilation controlled fire, rather than a perfect oxygen rich combustion should be considered. This is because in practice, majority of the fire will undergo various phases of combustions and availability of oxygen, and as shown in figures 3.11 and 3.12, the choice of a CO yield that is based on an oxygen rich ideal combustion does not match the experimental measurements.

Separately, we recommend future researches and design guidelines to outline whether a recommended CO yield value is based on ideal, oxygen rich combustion, and if not, the global equivalence ratio so that users of the CO yield can understand the conditions of combustion when selecting a CO yield.

Lastly, we have contacted Thomas Melcher, the corresponding author of the experiments [7] where we based on chapter on, and we did not receive a response.

3.5 Conclusion

This chapter shows, through the work by Melcher *et al* [7] that CO production has an inherent high variability, and this variability will carry into numerical simulations, and compounded by variability involved in the choice of CO yield to be used in simulations. Until more studies can be done to understand the repeatability of experiments for CO production, and the translation of the experiments into numerical models, engineers should avoid relying solely on CO to understand the tenability for occupants in a fire.

Because of the above, it is recommended for engineers to consider CO as a secondary variable when assessing tenability in a fire safety design. When used, sensitivity studies should be carried out to test a range of CO yields, as our research here has shown that the simulation results from a range of CO yield for a known material can have significant variability. When such a range of CO yield is available, engineers should consider using the higher range that better reflect the practical conditions such as a ventilation controlled fire, rather than an oxygen rich ideal combustion fire. If a material does not have a range, a factor of safety, e.g. 2 should be considered.

Consistent with the theme of this thesis, we strongly recommend that greater experiments and studies are carried out to consider the uncertainties associated with repeating experiments. Most often, many large and real scale fire tests that form the basis of numerical models are not repeated, i.e. the uncertainties associated with these experiments are unknown. To date, we have yet to see in our limited search extensive and recent peer reviewed studies that specifically report on, or consider the repeatability of the experiments, or the CO yield of various material properties.

As noted in a quote by Henry Petroski [37], 'Successful engineering is all about understanding how things break or fail'. It is of paramount importance for designers to understand the limitations of the tools used, in this case numerical modelling of CO.

Chapter 4

Unexpected Oscillations in Tunnel Fire Models

4.1 Physical Phenomenon or Numerical Artefact?

The starting point for this chapter traces back to the earlier work we did during my Master Thesis, whereby in the investigation of using CFD and FDS to carry out multi-scale modelling, we discovered significant oscillations in the mass flow (mixture of air and hot gases) out of the tunnel over time. Note this tunnel has an inlet and an outlet.

When we saw the oscillation, we wondered if it is a physical, or numerical artefact. There is evidence on the presence of pulsation effect in a fire [38], and the immediate question for us is how do we identify the source of the oscillation.

This chapter contains our work to understand if the oscillation is numerical or physical. If this is numerical, our work could help the developers understand the source of the oscillation, and to revise the code. If this is physical, it shows that the tunnel ventilation system would need to be designed for the top or max value of the mass flow in the oscillation.

Through this investigation, we identified the issue to be numerical. After numerous attempts in communicating this issue, we publicised this issue again in 2020 through Fire Technology [18], an internationally read journal for fire safety. Our attempts finally drew the right attention of the community, and we contributed to the developers [17] fixing this. The work here as allowed us in the tunnel fire modelling community to have greater confidence in the FDS solver for tunnel fires, and we are pleased to have contributed to the FDS community.

The work in this chapter has been adapted from material presented in FEMTC 2016 [39] and in Fire Technology [18].

4.2 Observation of Mass Flow Oscillation

Fire Dynamics Simulator [14] is commonly used in the practical application of tunnel fire safety design. When modelling fires using Fire Dynamics Simulator in long tunnels, we observed that mass flow out oscillates with a non-negligible amplitude of 20% or more. This phenomenon is only observed in tunnel fire models, and when a fire is introduced and is not observed in compartment fire models. This relates to the issue as discussed in Chapters 2 and 2.3.3 whereby a tunnel configuration is long but narrow, and the inherent incompressible flow setup of FDS means pressure travels instantaneously.

This oscillation is measured immediately at the tunnel outlet and before the end of the computational domain (10 m longitudinally from the tunnel's outlet). The mass flow is measured across the entire cross sectional area of the tunnel, and is a measurement of the bulk flow (mixture of air and hot gases).

When we first noted the oscillation, we noted engineers should not ignore the wiggles noting the lessons by Gresho *et al* [40] *"don't suppress the wiggles"*. Oscillation has not been reported in scientific literature or engineering reports as far as we know after an extensive search. This topic is rarely covered and we were unable to find any source discussing the presence of oscillation in the tunnels, including on the FDS forum or the advice of other specialists. It was only after our work has been published that Riess [10] published a technical report investigating the nature of oscillation in tunnel.

4.3 Oscillation in the Practical Application

In practical application, CFD, or computational fluid dynamics simulation is regularly used to test the ventilation design of a tunnel, both in normal operations and in fire emergencies scenarios.

The oscillatory behaviour in FDS matters in practical application because the results from the CFD models are used as part of the fire safety design. Engineers need to be able to understand if the tunnel ventilation system is behaving as designed in a fire, i.e. by sufficiently diluting smoke and heat from a fire, and ensuring smoke is exhausted from the tunnel in a way that supports the evacuation procedure.

If the mass flow, or other properties such as temperature and velocity [41] oscillates, engineers will not be able to determine if the parameters, such as temperature are within the tenable benchmark if these parameters oscillate by 20% or more at a frequency of approximately 4 s.

From a practical application's perspective, numerical oscillation is highly undesirable when carrying out a transient tunnel fire simulation where the tunnel's condition is assessed against fire growth over time. This is because depending on the point in time, the mass flow, and therefore other variables can be significantly under or over predicted, leading to false impression on the conditions in the tunnel. This oscillation means the toxic smoke experienced by evacuees would vary by 20% or more every 4 s and this would skew the results of a transient (time) based tenability assessment in engineering design.

With such significant oscillation in the parameters, it becomes unclear for engineers and designers to know if the tunnel ventilation is pushing an adequate volume of air, or if the dilution to the smoke and heat is adequate.

4.4 FDS Model Setup for the Tunnel

The tunnel used in the FDS model is largely inspired by the Dartford Tunnel in the UK where

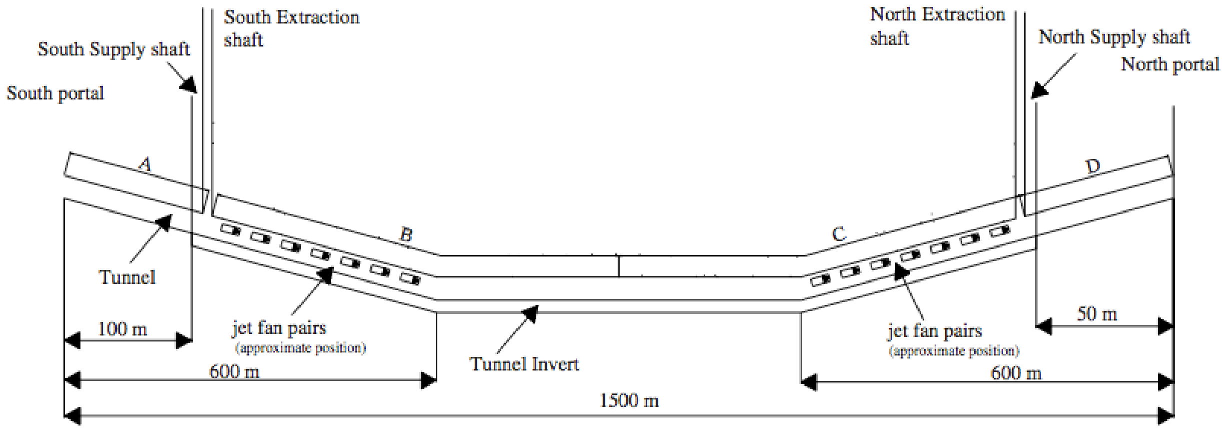


Figure 4.1: Long section of the Dartford West Tunnel [3].

the model uses the same length, cross sectional area and number of jet fans as the tunnel. It is not intended to fully replicate the details, such as the tunnel's curvature, elevation or the vertical ventilation shafts.

We have used FDS6.1.1, FDS6.5.3 and FDS6.6.0 plus the HVAC (Heating, Ventilation, Air Conditioning) feature to model the jet fans in the longitudinally ventilated tunnel. Firstly, a uniform grid size of 0.4 m is used in a single mesh. We have carried out a grid sensitivity study for the fire, and recorded consistent temperature output between the 0.2 m and 0.4 m grid.

As FDS is based on a rectilinear grid, the tunnel is represented as a straight duct of a square 6.4 m $\times$ 6.4 m (40.96 m^2) with no incline. On the basis this is a low Mach flow, we have approximated the tunnel as a square duct in FDS based on the tunnel's hydraulic diameter, $D_h = \frac{4A}{P}$, with A as the flow area, m^2 and P as the tunnel perimeter, m.

Although surface friction is a variable relevant to this study, to provide a representation of flow resistance, the tunnel walls are modelled as concrete with an absolute roughness, $\epsilon = 0.02\,m$ derived from a typical tunnel friction coefficient, $f = 0.026$ as the actual tunnel's properties are unknown. 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.

For the fire, the fire is located at 1 km in the tunnel. This distance of approximately halfway point of the tunnel is chosen so the fire is located far enough away from air stream from the jet fans. Further, at this distance the air flow in the tunnel is established, i.e. reducing the interactions between the jet fans' air flow and the fire.

A heptane fire with an area of 12 m^2 (4×3 m) is used. As this study is to consider flow oscillation, a simple heptane fire, rather than a solid fuel source such as polyurethane is chosen. With a simpler fuel source, this allows this study to be used as a benchmark in future experiments.

The fire area is kept consistent when we increased the fire sizes, as a different fire area dimension would result in a different volume of smoke that would affect the temperature of the smoke plume. We used three different HRRPUA (Heat Release Rate Per Unit Area) to achieve 35 MW at 2,920 kW/m^2, 55 MW at 4,590 kW/m^2 and 75 MW at 6,250 kW/m^2. This is the basis we selected a fire area of 12 m^2 that is between a typical passenger car and a small truck, to ensure the HRRPUA used is not unrealistically high. A radiative fraction of 0.35 was used.

The fire is set to achieve its full size 1 second after ignition with ignition at the 50 seconds mark. The 50 seconds mark is chosen to allow the flow to be established, and the 1 second ignition is chosen arbitrarily whereby we learnt later on it contributed, though not the sole cause of the oscillation.

Prior to the introduction of the HVAC feature, jet fans in FDS are modelled by specifying a fixed velocity boundary condition across the full area of the tunnel. This a rudimentary approach given the fixed boundary condition, the vent with the fixed velocity would be pushing a constant airflow and not accounting for the resistance generated by the fire as demonstrated by Vaitkevicius *et al* [42]. In practical application, the increased resistance due to the fire would need to be accounted for as this would mean larger fans are required, hence the need to model the jet fans as proposed here using the FDS HVAC feature.

Using the HVAC feature, a volumetric flow rate can be specified across two connected nodes thereby mimicking a jet fan's intake and outlet. 8.9 m^3/s is specified for each fan consistent with the tunnel specification and the fan start up is based on the FDS default instantaneous ramp up time. Note there are 14 jet fans pairs (28 fans) in total.

At the time of the work, FDS6.6.0 is the latest release. It is acknowledged at the writing of

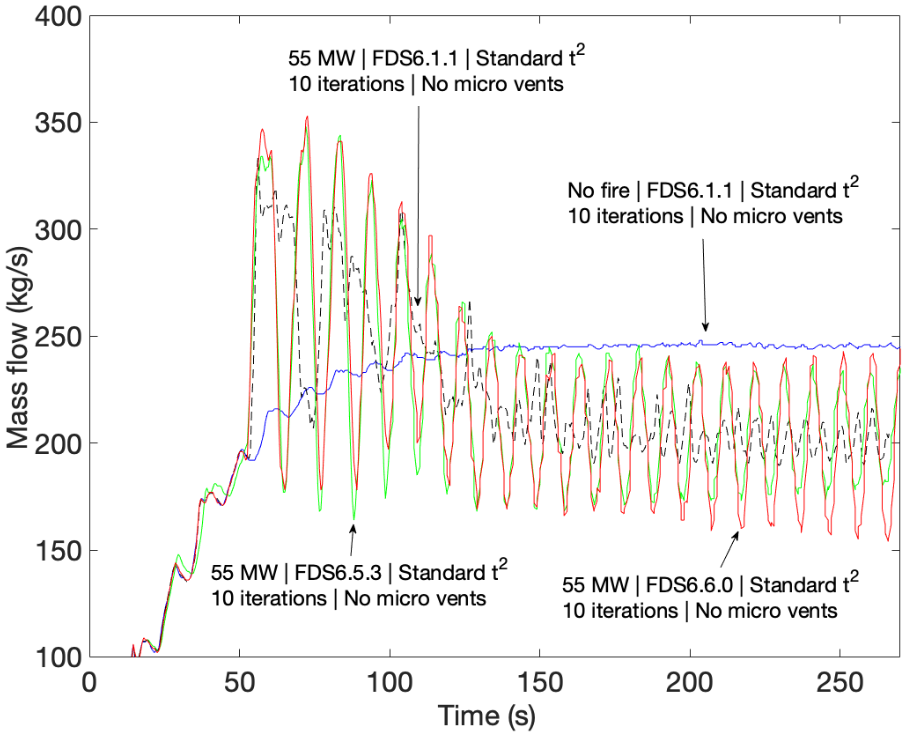


Figure 4.2: Comparison of mass flow out between cold flow (no fire) and 55 MW fire in three FDS versions.

this chapter there is a newer version of FDS. This is a similar issue in practical application whereby a project spanning a number of years would have been designed using an older version of a software, and the industry implicitly acknowledges the impracticality of re-assessing every simulations due to time constraints. This is on the basis the model was robustly implemented, and the exception is when major software flaws are discovered.

4.5 Investigating the Oscillation

For the simulation results, instead of velocity or temperature, we plotted the mass flow and its changes over time. This is because mass flow rate, m is a fundamental variable in the study of fluid mechanics that includes smoke movement. Compared to velocity or temperature that can be affected by other variables, tracking the mass flow in and out of the tunnel over time allows us to examine the fundamental flow behaviour.

As shown in Figure 4.2, in FDS6.5.3 and 6.6.0, the oscillation in the mass flow exhibits a periodic response whereas in FDS6.1.1, the oscillation is inconsistent, but with an average period of 10 s. Between FDS6.6.0 and FDS6.1.1, there has been changes that resulted in a different oscillatory behaviour in the mass flow.

For the investigation, a key question we wanted to answer is whether the mass flow oscillation as seen in the simulations is physical or an artefact of the fire model. We note Lonnermark *et al* [38] in the Runehamar tunnel large-scale fire tests observed pulsations in the velocity and temperature data recorded. A theoretical model has been used to study the oscillation phenomena registered during the large-scale fire tests in the Runehamar tunnel. Two different types of periods were seen, one near 4 s and one approximately 18 s.

Although the oscillation has been observed, there has not been an analytical model that accurately quantifies the oscillation as experimentally observed in tunnel fires. Our investigation attempted to reduce the number of possible causes that would numerically induce the oscillatory behaviour in an attempt to narrow down the scope of future investigations.

4.5.1 The Presence of a Fire

From figure 4.2, we compared the mass flows tracked in the fire simulations to the mass flows in a cold flow (no fire) scenario. We know the mass flows in and out of the tunnel in the cold flow scenario are relatively consistent without pronounced oscillations. Therefore, we know the oscillation only occurs when a fire is introduced to the tunnel, but only for a large fire, e.g. 20 or 30 MW and over.

Figure 4.3 plots the ratio of the mass flow at each complete oscillation, compared to the mean mass flow out of the tunnel, noting the mean value is between 150 to 500 seconds. A ratio greater than 1 shows the mass flow at each oscillation to be greater than the mean mass flow, whereas a ratio lower than 1 shows the mass flow at each oscillation to be less than the mean mass flow.In summary, any values outside of approximately 1 show the presence of oscillation and figure 4.3 shows for a small fire such as a 5 MW fire, there is no oscillation.

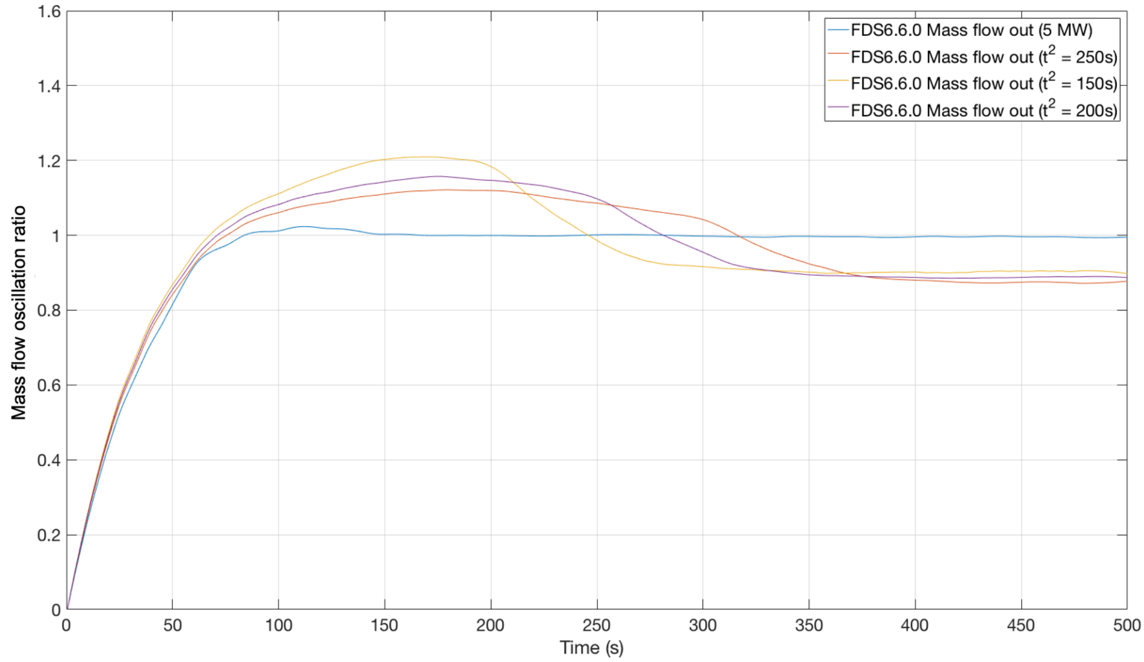


Figure 4.3: Examination of oscillation for a small 5 MW fire compared to 55 MW fires with three different fire growth rates. Any values outside of approximately 1 show the presence of oscillation either above, or below the mean mass flow out of the tunnel.

4.5.2 Discontinuity

In CFD modelling, if a change in a parameter is introduced abruptly into the model, discontinuity can occur. One of the major changes in the model is at the 50 seconds mark where the fire is introduced. This is a significant change as before 50 seconds, there was no heat source in the model. See Figure 4.4. We showed in Figure 4.4 that examples of oscillation can be seen in variables beside mass flow. Figure 4.4 shows depending on the fire growth curve, i.e. abrupt or smooth, there is a difference in the oscillation. This suggests discontinuity as a cause because if the oscillation in the fire size is physical, the magnitude and the frequency of the oscillation would be similar.

Given the fire itself is the main variable influencing the mass flow, to investigate this further, we carried out an additional simulation in FDS6.6.0 whereby we introduced a longer (smoother) t^2 fire growth curve to minimise the abrupt changes. In total, we modelled four different scenarios including a t^2 of 100, 150, 200 and 250 seconds for the fire to reach the peak 55 MW.

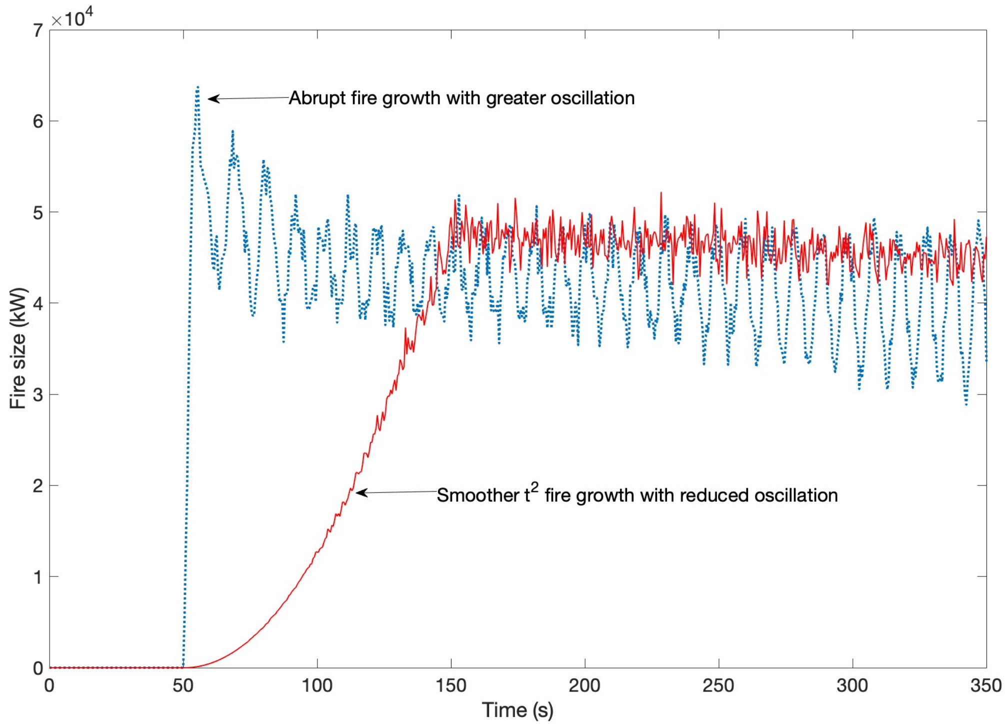


Figure 4.4: Comparison of oscillations between abrupt fire growth and smoother t^2 fire growth.

From Figure 4.5, the characteristics of the mass flow for all four scenarios behave differently, but at approximately 350 s onwards, the mass flow all oscillates around 200 kg/s.

Whilst the abrupt oscillation disappears when the fire growth curve is extended compared to the original case (Figure 4.2), the mass flow out of the tunnel continued to increase until the fire reaches the maximum size, i.e. the specified t^2 curve, where it then decreases to oscillate around 200 kg/s. At this point, we do not have an explanation or conjecture over this characteristics.

4.5.3 Changes in Mesh Sizes

Although we have carried out a grid sensitivity check, we have also simulated the 55 MW scenario using a finer 0.2 m grid. The rationale is that if the oscillation is physical, the oscillation observed in both the 0.2 m mesh and 0.4 m mesh model would be identical. As shown in Figure 4.6a, the oscillatory profile in the models with different mesh sizes differ. At this point, we understand by extending the t^2 fire growth curve or by changing the grid size the major

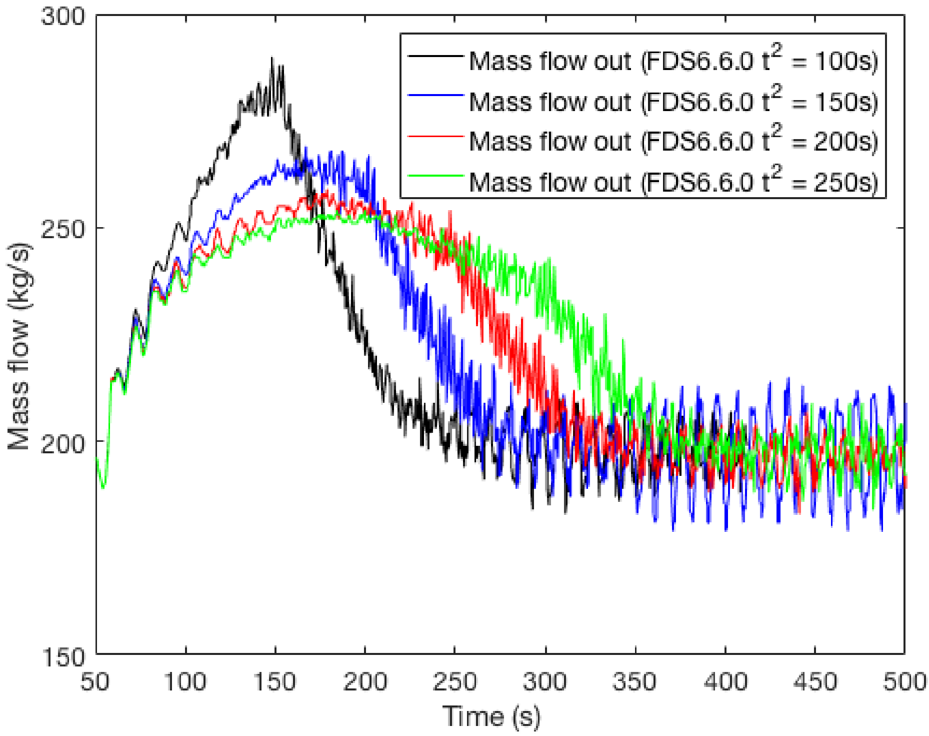


Figure 4.5: Comparison of mass flow out of the tunnel for four different 55 MW fire growth curves.

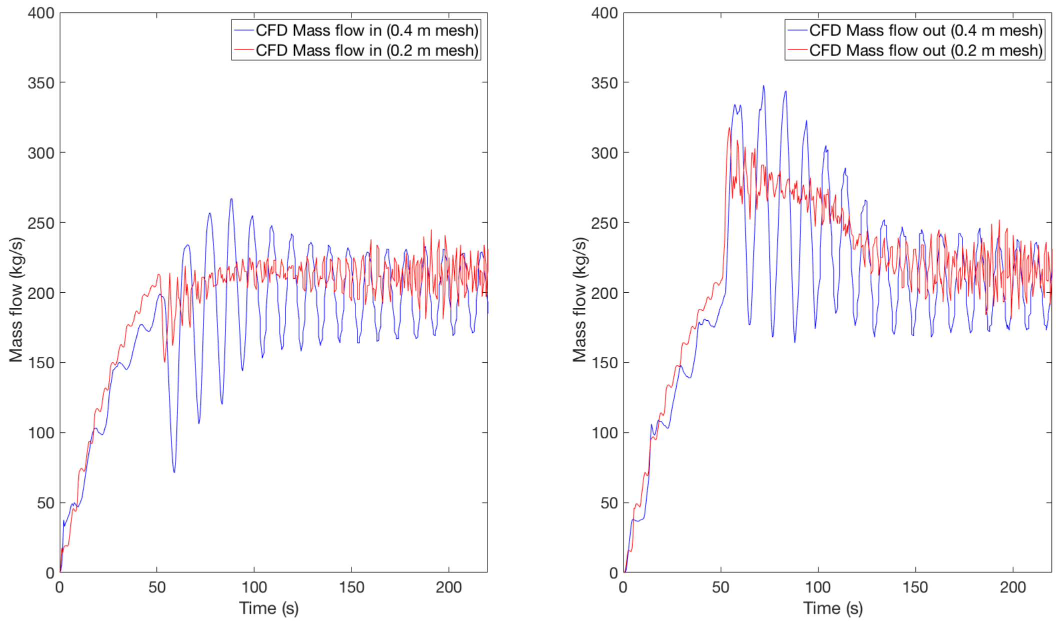
oscillatory behaviour changes. Whilst not conclusive, this reduces the confidence the observed oscillation in the model is physical.

4.5.4 FDS Pressure Solver

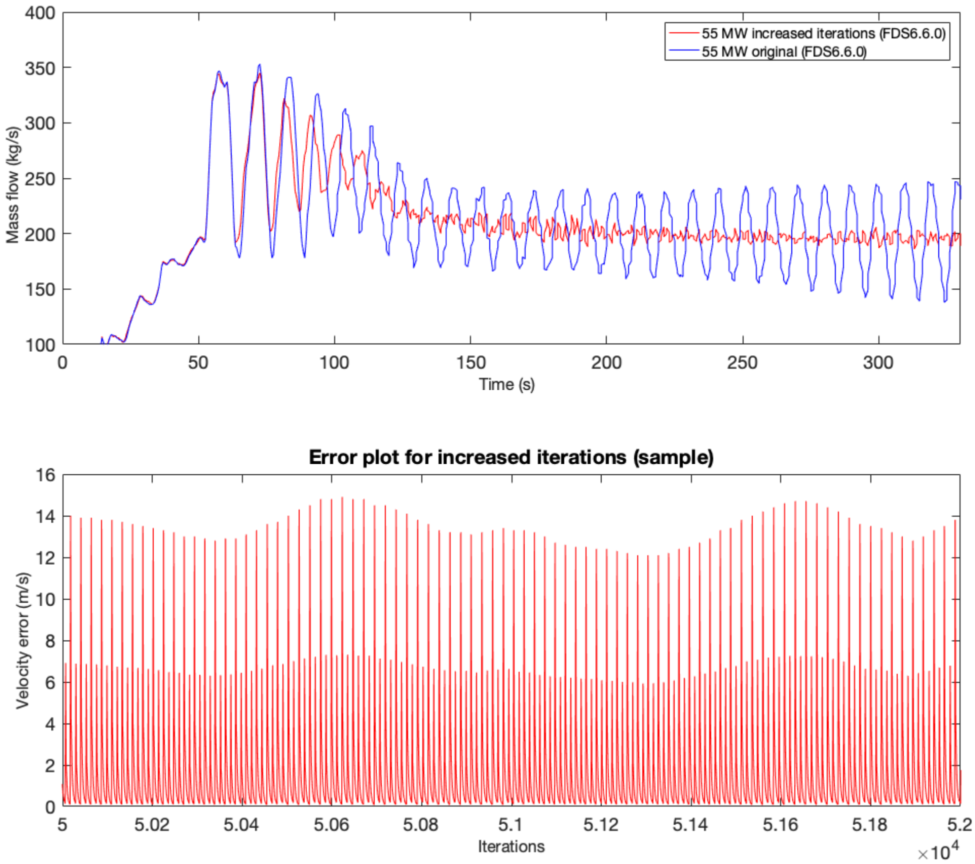
Per the FDS Guide [14] Section 6.6 including Section 6.6.2, the FDS Guide has a brief section on fire modelling in long tunnels. One common source of numerical instability noted is the use of multiple meshes, and the interface between these meshes in the model. Given all the models are based on a single mesh with extended domain beyond the portals, this common source of numerical instability is omitted.

The FDS Guide notes a number of configurations for the pressure solver to tightened the error tolerances by increasing the maximum number of iterations (default is 10 for FDS6.6.0).

From Figure 4.6b, we know by increasing the iterations of the pressure solver to 30, the oscilla-



(a)



(b)

Figure 4.6: 4.6a) Comparison of mass flow in and out between 0.2 and 0.4 m grid size. 4.6b) Comparison of mass flow out between default and increased pressure solver iterations. Mass flow oscillation became negligible, but the velocity error continues to oscillate.

tion first appeared at 50 seconds gradually reduces. Although 30 iterations were specified, the pressure solver only computed up to 14 iterations. In addition, although the oscillation in the mass flow became negligible with increasing iterations, the velocity error continues to oscillate with large differences at 0 to 14 m/s between small timesteps rather than lower differences in longer timesteps, indicating there is an underlying numerical issue with the pressure solver.

We have not attempted to tighten the error tolerance of $\delta_x/2$ (δ_x is the grid size) due to the impractical increased in computational cost from a practical application perspective, and that we have considered the impact of grid sizes on the oscillations.

4.5.5 Micro-vents

The tunnel is modelled to be airtight with no pressure relief other than the portal. This airtightness resulted in unrealistic pressure causing large fluctuations stressing the FDS pressure solver. Discussion with McDermott [43] suggested introducing vents along the tunnel to simulate leakage to reduce the stress on the pressure solver.

We have coined the vents noted here as micro-vents as they are one cell sized, and located along the tunnel and the location of the fire. Micro-vent is used to separate this term for vents such as actual tunnel exhaust vents that are present in a real tunnel.

As shown in Figure 4.7, the magnitude of the oscillation became negligible by using 4 micro vents of 1 cell size at atmospheric pressure along the location where the 55 MW fire is located. Although micro vents reduced the mass flow oscillation to negligible, the velocity error is still oscillating and increasing over time. The vents are unrealistic as a micro sized vent is not present along a tunnel, and not all real fire scenarios would have the fire placed near a vent.

Modelling a large fire in a long and confined tube is an interesting computational problem due to the pressure expected, and in theory the tunnel domain is modelled to be airtight with no pressure relief other than through the portal. This artificial airtightness resulted in unrealistic pressure causing large fluctuations that stress the pressure solver in FDS.

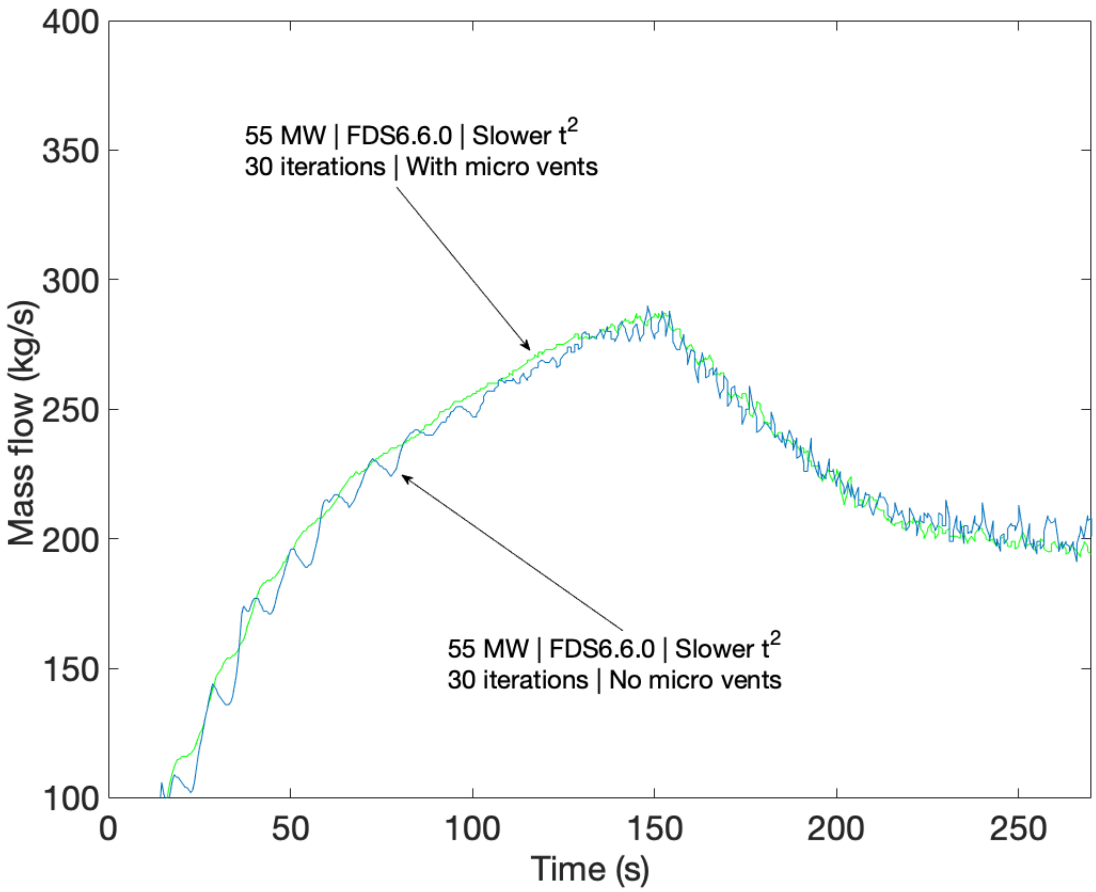


Figure 4.7: Significant reduction in the magnitude of the oscillation if micro vents are introduced.

Adding these vents alters the solid obstruction condition, and provides a pressure relief or pressure flucutation altering the condition of the pressure solver. The vents are unrealistic in reality as a micro sized vent will not be practical in removing any substantial smoke or heat. However, this numerical hack can noticeably decrease the occurrence of oscillation.

The use of micro-vents has been subsequently introduced into the official FDS documentation [14]. From a practical application perspective, the use of micro-vents as noted earlier is not considered to reflect reality. Whilst leakages in tunnel occur, for example near the cross passages doors, these leakages are not always conveniently located near the location of the fire and therefore should not be relied upon as a numerical solution.

4.5.6 Our Findings

With the investigation, we firstly conclude there is negligible oscillation for a small fire. The oscillation occurs when there is a larger fire size, i.e. 30 MW or greater.

Secondly, for the original scenario where the fire starts immediately at 50 s and reaches the peak heat release rate within 2 seconds, the oscillation increased over time, signifying a blow up, i.e. numerical error. We know this is due to the setup of the fire that introduces shock to the numerical scheme. Once the fire growth is stretched out, the mass flow converges over time.

Thirdly, whilst the mass flow for all the modified scenarios converges, there are still non-negligible oscillation.

So far, we know the nature of the oscillation is affected by the setup of the model, namely fire size, fire growth curve and pressure solver configuration. The results due to increasing the pressure solver iterations, and using a finer mesh sizes point to the oscillation being a numerical artefact, in particular because the period changes from negligible, 4s and 8s for the same fire scenarios. For a physical issue, we do not expect major changes by increasing the fidelity of the pressure solver or the mesh sizes.

In FDS version 6.7 whereby the FDS developers fixed the numerical oscillation [17] following the publication of our paper [18], recent versions (at the time this chapter was written) has also introduced a new feature to the solver. Previously due to the exchange of information at the mesh boundary, a good practice for simulating fires in a long tunnel is to avoid separating the tunnel into multiple meshes.

Whilst multiple meshes speed up the computational time, the downside is the numerical instability and the numerical errors that could be introduced. In FDS version 6.7 onwards, a new feature noted as PRECONDITIONER [14] has been introduced. The purpose of this feature is two-fold, it is firstly to mitigate the numerical oscillatory behaviour observed, and secondly it is to enable the use of multiple meshes. This feature has been considered in the subsequent chapter 5 where FDS is used to investigate the presence of fire throttling effect in tunnel.

The technical implementation of PRECONDITIONER is explained in the FDS technical reference [14]. Briefly, PRECONDITIONER is based on solving a global 1D pressure equation across the entire tunnel, which is then added to the 3D pressure fields across the multiple meshes. This implementation isolates the significant pressure fluctuations at the fire, and as a result reduces the complexities for solving the 3D pressure fields as there are less significant pressure fluctuations. This means the combination of a 1D global pressure solver, coupled with the 3D pressure fields solver speeds up the convergence of the overall 3D pressure fields across the entire tunnel.

Although in theory either micro-vent or PRECONDITIONER could be used to mitigate the numerical oscillation, from a practical application perspective engineers should consider the use of PRECONDITIONER given this better reflects the true condition of a tunnel, and enables the use of multiple meshes.

Lastly, when we were carrying out the investigation for the oscillation effect, we identified the lack of best practice guidelines in tunnel fire modelling. This includes various boundary conditions to simulate a networked tunnel, obstruction, surface conditions, inclination, large fires, wind effects and others that could further complicate the model. It took a long time to find the solution and most engineering projects do not have the time or resources to devote so much time to this type of issues.

We can learn from other industries, for example the Architectural Institute of Japan [44] published a set of guidelines for the practical application of CFD to pedestrian wind environment around buildings or the US Nuclear Energy Agency [45] best practice CFD guidelines. Although there is some guidance in FDS User Guide [14], a broader guideline similar to those in the industries would be beneficial. The guideline would provide a good practice CFD modelling recommendations when using CFD to assess tunnel fire safety design. Chapter 2 of this thesis is our first attempt at highlighting specific practices to be considered when modelling fires in long tunnels using FDS.

4.6 Conclusions

With a lack of real tunnel fire experiments in which CFD models can be verified and validated against, this chapter shows the importance of digging deeper into the results and the code for a seemingly inconspicuous issue. When we first noticed and highlighted this issue, it was a challenge communicating and attracting attention to this issue.

This chapter reflects the sustained effort we made to get to the bottom of the issue, and our effort in communicating this issue over 5 years since we first observed the issue. We managed to get to the bottom of the issue, and through the great effort of the FDS developers, the tunnel ventilation community now has a revised version of the FDS code.

In this chapter, we have re-run the models using the revised FDS version where PRECONDITIONER is included to minimise the occurrence of unexpected oscillations in the tunnel. This is the case where there is a significant change in the code that directly impacts our work. Compared to the work in Chapter 3 where the fire is located in a square room, and that there has not been fundamental changes to the species (CO) modelling that we have not re-run the model. This is a practical example where engineers need to consider on a case by case basis whether a re-run is required.

Through our work above, it means today, if we see oscillations when modelling a tunnel fire, it may actually be showing a physical phenomenon such as the pulsation effect [24]. Further, the work in this chapter gives us the confidence the conclusion we are drawing in the next chapter.

In our journey to publicise this observation, we have learnt it is important for designers to consider the phenomenon of oscillation for a large fire in a long tunnel needs to be considered at all times and to determine whether the phenomenon will impact the safety outcome of the design. To paraphrase Gresho and Lee [40], ‘Don’t ignore the wiggles’.

Chapter 5

Fire Throttling

5.1 Understanding Fire Throttling

The aim of this thesis is to consider the role of numerical simulation in engineering design. As noted in the earlier chapters, we have investigated the viability of using CFD, specifically FDS, to model tunnel fires in practical application.

Whilst we have identified several limitations with FDS, FDS remains one of the most widely used solvers in fire safety design. Working within these limitations, the aim of this chapter is two-fold: Firstly to understand the fire throttling effect in long tunnels, a phenomenon that has been mentioned but to date has yet to be accurately described, and secondly to show that CFD and FDS remain an important tool in the design of tunnels.

This chapter will show through 1D and CFD methods that for larger fire sizes, CFD is essential to capture the 3D effect of a fire that otherwise cannot be (at this stage of our boundary of knowledge) captured in a 1D model. This shows, CFD and FDS, if used within the limitations considered in the earlier chapters, remains a critical tool in design.

This chapter contains a literature review of the fire throttling effect, followed by a discussion of its importance in practical applications, and an investigation of a 1D model to describe fire throttling.

The work in this chapter has been published in Fire Technology [21]. It has also been presented at the Asia-Oceania Symposium for Fire Science and Technology [20].

5.2 Tunnel Ventilation and the Fire Throttling Effect

In most tunnels, longitudinal ventilation is preferred to transverse systems due to their comparatively easier design and construction. Longitudinal ventilation can be achieved via jet fans, ventilation shafts or nozzles providing significantly simpler construction compared to extended plenums and smoke ducts for transverse system. The design of a longitudinal ventilation system is a well studied topic which includes design fires, critical ventilation velocity and back-layering [5, 46].

In a longitudinally ventilated tunnel, the ventilation principle in a fire is to push air along the tunnel, usually in the direction of traffic travel so the upstream of the fire is kept free of smoke. See figure 5.1. When designing such systems, the various contributors of pressure losses, such as surface friction, traffic obstructions, tunnel geometry and incline need to be consider. Fire is a key influence in the design of longitudinal ventilation systems.

The size of a fire, and its influence in the airflow velocity required to create a smoke free area upstream of the fire is the well studied topic of critical ventilation velocity. Further, it has been known for over 40 years [47] that a fire in a closed duct, e.g. a tunnel, results in pressure loss. The studies showed that the presence of a fire has a similar effect to that of a narrowing, or blockage, at the location of the fire. The terms *fire throttling* or *fire choking* used to refer to this effect may originate from the terminology employed to describe similar aerodynamic behaviour observed in internal flows within ducts and nozzles.

This issue was first discovered over 40 years ago in research on mine tunnels by Greuer *et al.* [47] and over the last 20 years, this issue has been detailed in various publications [25, 9, 8, 48]. In the next sections, we set out to provide a chronological summary of the key researches for the past 40 years since the first scientific report by Greuer [47]

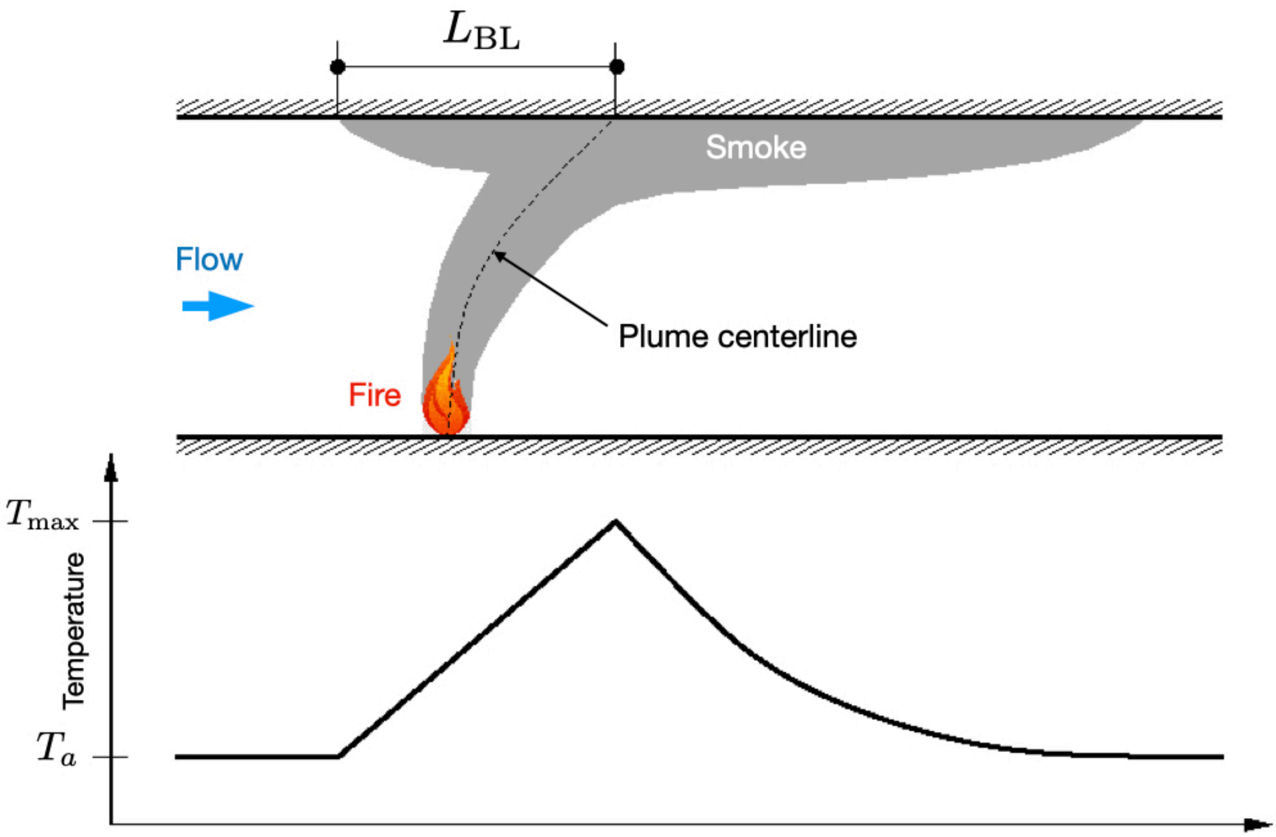


Figure 5.1: A tunnel fire in a longitudinally ventilated enclosed tunnel: notation (top) and an indicative steady-state temperature distribution (bottom). The distance L_{BL} represents the length of backlayering, when it is present and the region where the average temperature (for illustrative purposes only) increases from ambient to its maximum value. Note that in the following we will assume the maximum temperature is a point source, with $L_{BL} = 0$.

5.3 A History in the Research of Fire Throttling Effect

5.3.1 First report by Greuer

In 1973, Greuer [47] made the first scientific reporting of the fire throttling effect in the investigation of mine fires on the ventilation of underground mines. At the time it was first observed that the prediction of the airflow in a mine tunnel can be complicated by a fire due to the forces generated by the fire that could disturb the smoke plume. Although the research focuses on the properties and the temperature of the fire itself in the mine tunnel, Greuer provided insights to understand the forces developed by the smoke plume.

Greuer [47] suggested that when a fire is introduced in a tunnel, the presence of a fire results in additional head loss and pressure loss. Greuer [47] noted that head loss describes the dissipation of energy in a system due to the resistance it encounters, for example friction, and it is described in per unit weight of air. Similarly pressure loss occurs due to the resistance the fluid flow encounters, but pressure loss is described in per unit volume of air. The pressure loss and head loss can be related using the specific weight, and Greuer noted that in a ventilation system the specific weight can undergo material changes and therefore the head loss and pressure loss for the throttling effect needs to be considered separately.

Greuer suggested in the research that the fire throttling effect principally occurs when the air flow passes through the vicinity where the fire is. Greuer noted the occurrence of two key effects here affecting the air flow. That is firstly: mass flow increase (injection of additional mass) due to the addition of combustion products and evaporated water; secondly the additional pressure loss caused by the increase in temperature.

In reference [47] , Greuer made a clear distinction between pressure loss and head loss. Simplistically, the key difference is that pressure loss is expressed in the volume of air, whereas head loss is expressed in the mass of air. Describing the loss using Darcy's equation, and by assuming constant friction, cross section area and the pressure generated by the fire, the researcher [47] postulated that the additional losses (throttling effect) are described by a factor:

$$\frac{T_m}{T_a} - 1 \tag{5.1}$$

where T_m is the average temperature in the vicinity of the fire, and T_a is the ambient air temperature. The factor indicates that an increase in fire temperature leads to an increase in the resistance to the flow (throttling effect).

The reader is referred to [47] and for the derivation of this equation. The key takeaway here is that Greuer [47] suggested at the time that temperature is a reasonable predictor in the occurence and magnitude of the fire throttling effect.

5.3.2 Hwang and Chaiken

Following the work by Greuer [47] in 1973, the second key publication in the topic of the fire throttling effect is in the 1978 where Hwang and Chaiken [49] published their research on the effect of a duct fire on the velocity of the air ventilation. Essentially, Hwang and Chaiken set out to analyse the air velocity in the presence of a fire, and the air velocity in the absence of a fire.

A key consideration made by Hwang and Chaiken [49] is that the fire is to be localised, i.e. it has a short axial length. This is similar to the type of vehicle fires, either a train carriage or a road vehicle found in a tunnel. This is also a consistent consideration as suggested by Greuer [47].

This consideration is important in the control volume analysis carried out by Hwang and Chaiken [49] where the fire is expected to increase the air temperature in the vicinity of the fire. This enables the assumption that the temperature in the control volume is uniformly heated. This is a reasonable assumption when the fire is localised, otherwise a fire with a longer base means that in reality the air temperature across the longer fire base is expected to vary.

Hwang and Chaiken [49] proposed that the throttling effect due to a fire can be approximated by:

$$\frac{V_c}{V_\infty} = (1 + M_o)\frac{T_h}{T_\infty} \tag{5.2}$$

where V_c is the velocity downstream of the fire, V_∞ is the velocity upstream of the fire, M_o is the mass injection factor, T_h is the temperature downstream of the fire and T_∞ is the temperature upstream of the fire (ambient).

Equation (5.8) suggests the same as proposed by Greuer [47] in that the increase in temperature influences (increases) the resistance to the flow, i.e. the throttling effect. However, Hwang and Chaiken [49] added that an increase in the mass injection (a larger fire) into the tunnel will also influence the air flow due to the throttling effect.

The derivation of equation (5.8) by Hwang and Chaiken [49] relied on several key assumptions. Firstly, the flow in the tunnel is assumed to be steady (bulk flow) and one dimensional only with no stratified flow. Secondly as mentioned earlier, the fire is assumed to be localised, i.e. the gas temperature changes from T_∞ to T_h in a short distance. Thirdly, the tunnel geometry, and the fire are assumed to be constant.

Using the above assumptions, Hwang and Chaiken [49] suggested that there is a relationship between the ventilation velocity in a fire to the temperature differences and the mass injection factor M_o, which are also influenced by the tunnel geometry and ventilation conditions.

Although the research proposed is informative, it does not lend itself for practical application. This is because M_o, a key value in the equation, is a parameter that is an experimental value [49]. This means the value cannot be readily derived and from a practical application perspective is therefore impractical. This is also the reason why the equation was not compared in the studies at the later section due to the inability to derive this value from a practical application perspective, e.g. a CFD study will need to be carried out to approximate this value, which would then negate the use of a 1D model.

5.3.3 Experiments Related to the Throttling Effect

At this juncture since the works by Greuer [47] and Hwang and Chaiken [49], a key question that we asked, and many would also ask is whether the earlier works have been experimentally assessed. In our review, we identified two experimentally based research papers by Lee *et al* [50] in 1979, and by Litton *et al* [51] in 1987.

The pressure loss across the fire zone (throttling effect) is due to three main factors: the injection of mass owing to the combustion, the elevated temperature due to the combustion, and viscous dissipation in the turbulent mixing of the hot gases. They noted the cracks in the timber burnt in the experiment caused a pressure loss in the flow which is ignored. This is because the loss is due to the surface condition of the material burning, and is not a fundamental factor, i.e. the throttling effect is expected to occur even without any cracks or losses on a material burning, or if it is a different material burning, e.g. oil based pool fire. They finally proposed that the fire throttling loss across the fire can be represented by a standard pressure loss equation:

$$\Delta P = f\frac{\rho V^2 L}{2D} \tag{5.3}$$

Lee *et al* [50] noted that the first two causes can be represented by ρ (density) and V (velocity), and the third factor is represented by f (friction factor). These values used in the experiments were experimentally derived.

Following the work by Lee *et al* [50], Litton *et al* [51] in 1987 carried out a medium scale and a large scale experiment to investigate the effects of fires on mine tunnel ventilation, with specific references made to fire throttling.

5.3.4 The Revived Interest in Fire Throttling

Since the research by Litton *et al* [51], the topic of fire throttling effect has largely been covered infrequently. Figure 5.2 shows a snapshot in time of the key researches related to fire throttling.

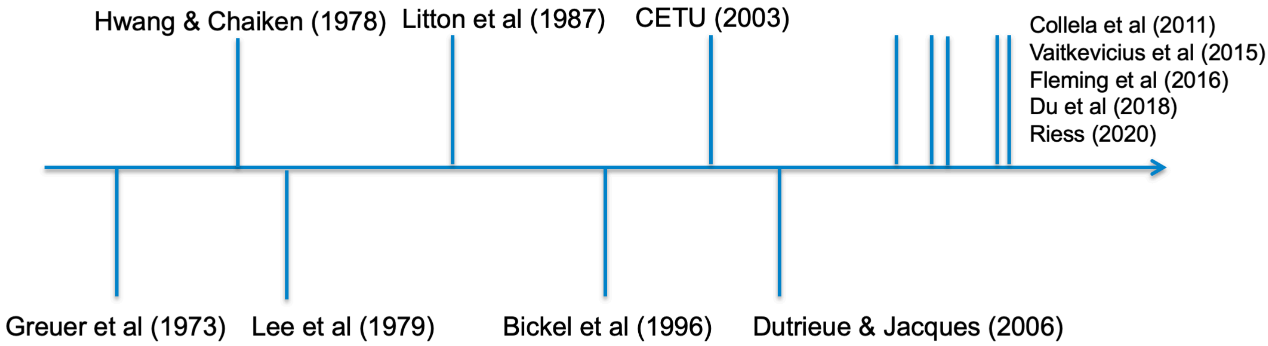


Figure 5.2: A snapshot in time of the key researches related to fire throttling

Over the last 20 years, this issue has been detailed in various publications [25, 9, 8, 48], and despite renewed interests in recent years [42, 10, 52], the literature describing this effect, including the understanding remains scattered and inconsistent. There have been several equations proposed to treat the fire throttling effect, and two of such equations are from CETU [9] and Dutrieue and Jacques [8] that consider the pressure drop directly at the fire, but not downstream of the fire. These two equations are examined in later sections, namely equations (5.31–5.32).

Despite this, it is acknowledged at a fundamental level in the various publications [25, 9, 8, 48], and certainly in design considerations [53], that the occurrence of the fire throttling effect has a material impact in practical applications.

This is because the increase in flow resistance due to the fire requires extra fans to achieve the required critical ventilation velocity [42]. Whilst there is a safety buffer in the sizing of the tunnel ventilation system by oversizing the fan capacity, this is typically in the region of 5% fan capacity. This represents a volumetric increase of 3 to 8 m^3/s in volume flow rate for a typical tunnel, and is inadequate to deal with any major losses that are not already accounted for in the design. Engineers and tunnel designers will know it is impractical to simply increase the fan capacity by a larger margin because this would impact the sizing of the ventilation shaft in tunnel, and therefore the tunnel diameter, i.e. tunnelling cost. In a project, the sizing of the tunnel's diameter can make or break the capital expenditure of a project.

In this research, we aim to amalgamate the conjectures published to date, and distill the fire throttling effect into a concise description. This description is presented in a one-dimensional model based on pipe flow principles that aims to incorporate the effects of a fire. We achieve this in the one-dimensional model by identifying the zones in the tunnel impacted by the fire, and proposing a specific model for each of these zones.

Using this description, we then evaluate their contribution to the fire throttling effect, and this is achieved by comparing the pressure losses calculated by the one-dimensional model to those obtained via CFD simulations. CFD simulations are chosen to verify the one-dimensional model due to the lack of experimental data obtained from simultaneous measurements of pressure and temperature that would be required to validate it.

Our secondary objective is to provide a description of the fire throttling effect that forms the foundation for practical application, or for further research. This is one reason why the one-dimensional model is based on pipe flow principles, i.e. the same principles routinely used in the analysis and design of longitudinally ventilated tunnels.

5.4 1D Modelling of the Fire Throttling Effect

Since the fire throttling effect is characterised by pressure losses, we will describe in this section a 1D model, based on the standard assumptions of pipe flow engineering theory [54], to calculate the pressure losses due to the fire in terms of the temperature distribution it generates along the tunnel.

For the sake of simplicity and ease of analysis, the modelling scenario is that of a single fire in a straight enclosed tunnel of constant section with only two openings: the entrance and exit portals. Flow quantities in the 1D model, such as flow velocity and pressure, are based on area-averaged values. As it is customary in these 1D models [46], we assume the momentum balance of the flow in the tunnel to be the sum of independent contributions of the form

$$\sum_k \Delta p_k = 0 \tag{5.4}$$

where Δp_k represent pressure changes due to the various effects considered in the model. Terms of this form have been proposed to model the effect of wind, obstacles, wall friction, buoyancy, fans, inflow and outflow portals, and tunnel inclination amongst others.

This chapter will focus on the terms that are the main contributors to pressure losses in a longitudinally ventilated tunnel in the presence of a fire. We will present and discuss formulations for those terms leading to the throttling effect, and assess their suitability by comparison with CFD results. In what follows we will consider only three independent contributions to the total pressure losses, Δp, namely

$$\Delta p = \Delta p_a + \Delta p_f + \Delta p_s \tag{5.5}$$

where the terms in the right-hand side denote the pressure losses in the cold air zone of the tunnel, Δp_a, the local pressure losses at the fire site, Δp_f, and the pressure losses in the zone filled with hot smoke downstream of the fire, Δp_s. The subindices a, f and s are shorthand for air, flame, and smoke, respectively. Δp_a and Δp_s are pressure losses distributed along the length of the tunnel whereas Δp_f is a localised pressure loss at the site of the fire.

5.5 Calculation of Pressure Losses

The calculation of the pressure losses in equation (5.5) will utilize the notation of figure 5.3 in the following sections. This figure shows the domain split into three zones: a zone of cold air at the entrance of the tunnel upstream of the fire where we will assume the values of the flow variables are constant, a zone in the vicinity of the flame that will be assumed to be of negligible length compared to the length of the tunnel, and a zone of hot smoke downstream of the fire where its influence will be felt.

As stated in the previous section, our aim is to calculate the pressure losses corresponding to these three zones, i.e. Δp_a, Δp_f, and Δp_s, respectively.

The distributed pressure losses in the tunnel contributing to the throttling effect are calculated through the pressure gradient calculated via the Darcy-Weisbach friction coefficient, λ, using

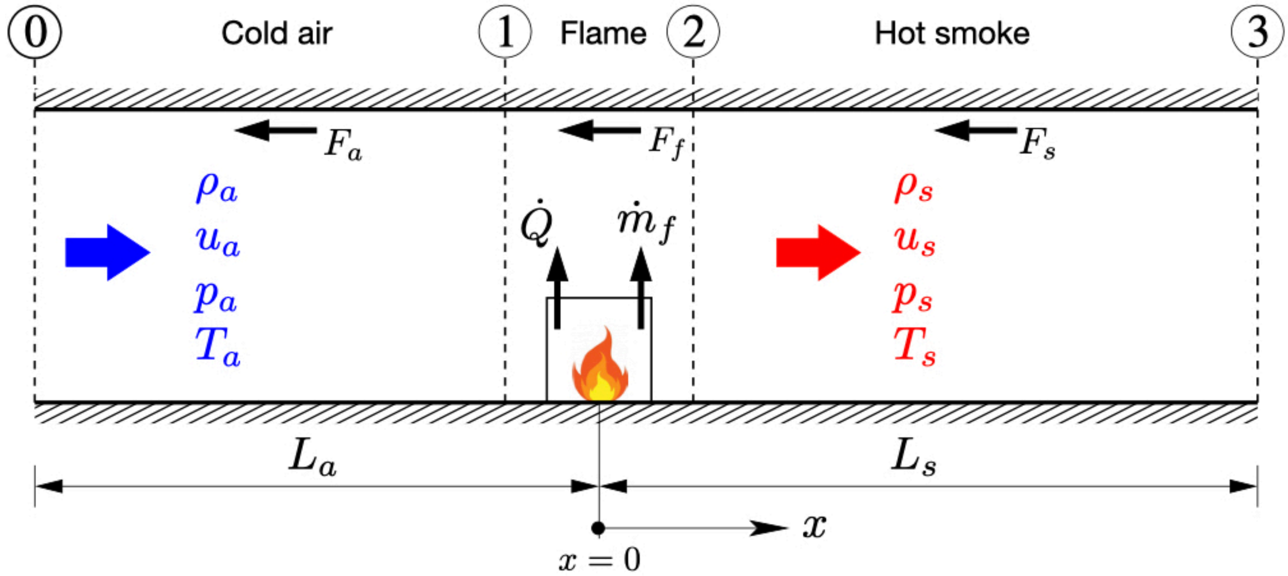


Figure 5.3: Notation for the calculation of the pressure losses.

the formula

$$\frac{\mathrm{d}p}{\mathrm{d}x} = \frac{1}{2}\frac{\lambda}{D_h}\rho u^2 \tag{5.6}$$

where D_h is the hydraulic diameter of the tunnel cross section, and the friction coefficient $\lambda(\mathrm{Re})$ is given a function of the Reynolds number, Re, via the commonly used pipe flow engineering formulas [54]. Here we have adopted the explicit approximation to Colebrook's formula for the friction coefficient [55], namely

$$\lambda = \left\{ 2\log\left(\frac{\varepsilon}{3.715} + \frac{5.72}{\mathrm{Re}^{0.9}}\right)\right\}^{-2} \quad ; \quad 65 < \varepsilon\,\mathrm{Re} < 1300 \tag{5.7}$$

where ε is the relative surface roughness in the Moody diagram.

The additional distributed friction generated in the hot smoke zone is accounted for through changes in velocity due to temperature as proposed in references [51, 56]. For an enclosed tunnel of constant cross-sectional area, temperature corrections are obtained by a combination of the continuity and the perfect gas equations leading to

$$\rho u A = \text{cons.} \quad \& \quad \rho T = \text{cons.} \quad \Longrightarrow \quad \frac{u}{T} = \text{cons.} \tag{5.8}$$

Therefore, if the distribution of temperature, $T(x)$, along the tunnel is known, equation (5.8) will be used to update densities and velocities according to

$$u(x) = u_a \frac{T(x)}{T_a}; \qquad \rho(x) = \rho_a \frac{T_a}{T(x)} \tag{5.9}$$

where the subindex a indicates again the (constant) values in the cold air zone in the tunnel. The form of the temperature distribution adopted in this work is described in the following section.

5.5.1 Temperature Distribution Along the Tunnel

As noted earlier, this calculation method for evaluating the pressure losses due to a fire relies upon specifying a temperature distribution. This section describes the temperature distribution adopted here.

Let x denote the coordinate along the length of the tunnel, in the direction of the incoming flow of air, with the fire source located at $x = 0$. We assume the minimum flow velocity is equivalent or greater than the critical ventilation velocity which allows us to neglect back-layering and plume deflection effects so that we can assume a temperature distribution of the form

$$T(x) = \begin{cases} T_a & x < 0 \\ T_s(x) & x \geq 0 \end{cases} \tag{5.10}$$

where T_a represents the (constant) temperature in the cold air zone upstream of the fire, and $T_s(x)$ represents the variation of temperature in the hot smoke zone downstream of the fire. The adopted form of $T_s(x)$ will be discussed in Section 5.5.4. Here we are assuming that the presence of the fire leads to a jump in temperature at $x = 0$ with $\Delta T = T_s(0) - T_a > 0$. The value of ΔT is a function of the fire heat release rate (HRR).

Note that, if required, the effects of back-layering and plume deflection could be incorporated in the temperature distribution through empirical formulas such as those proposed in references [57] and [58], respectively. For now, we will proceed on the basis there is negligible back-layering.

We will now proceed to describe how to calculate the three contributions to the pressure losses given such a temperature distribution.

5.5.2 Pressure Losses in the Cold Air Zone

Following the notation of figure 5.3 and using the assumed temperature distribution, the pressure losses, Δp_a, in the cold air zone $(0 \to 1)$ are obtained by integrating the pressure gradient given by equation (5.6) with constant temperature, T_a, and therefore constant velocity. The pressure losses Δp_a are given by

$$\Delta p_a = \frac{\lambda_a L_a}{2D_h} \rho_a u_a^2 \tag{5.11}$$

where λ_a denotes the constant friction coefficient in the cold air zone. In the comparison with CFD results that will follow in the verification of the method to be presented in Section 5.7, the value of the surface roughness ε in equation (5.7) is selected to match the (constant) pressure gradient of the computed solution in the cold air zone. This calibration ensures the friction due to cold air flow is the same in the 1D model and the CFD simulation. Wewe recognise there may be uncertainties in the CFD models, and this calibration mitigates this because by matching the friction in the cold region between the 1D model and the CFD simulation, we have certainty any discrepancy is due to the temperature only.

5.5.3 Pressure Losses Across the Fire

We obtain two alternative expressions of the localised pressure losses caused by the fire, Δp_f, through a conservation analysis using the control volume in the flame zone $(1 \to 2)$ shown in figure 5.3. The first one relates Δp_f to the HRR of the fire, $\dot{Q}$, and the second one to the temperature jump across the fire. This temperature jump can be obtained from semi-empirical formulas such as the plume theory which will be described in detail later in this section. The subindex 2 in the following derivations will denote the values immediately upstream of the flame zone which we recall it is assumed to be of zero length. For instance, $\rho_2 = \rho_s(0)$ where $\rho_s(x)$ is the density in the hot smoke zone as illustrated in figure 5.3.

Denoting by $\dot{m}_f$ the mass flow produced by the fire, conservation of mass, or continuity, states that

$$A_T(\rho_a u_a - \rho_2 u_2) + \dot{m}_f = 0 \tag{5.12}$$

where A_T is the (constant) cross-sectional area of the tunnel. Balance of momentum can be expressed as

$$A_T(p_2 - p_a) + F_f = A_T(\rho_a u_2^2 - \rho_2 u_a^2) \tag{5.13}$$

where F_f is the pressure loss due to friction at the fire site. Finally the conservation of energy reads

$$\dot{Q} + \dot{m}_f c_p T_a = A_T c_p(\rho_2 u_2 T_2 - \rho_a u_a T_a) \tag{5.14}$$

where c_p is the specific heat capacity at constant pressure. Note in later sections we will only consider the contribution of the convective energy to the HRR $\dot{Q}$ following reference [46]. We neglect fire mass flow, i.e. $\dot{m}_f \approx 0$, with the assumption that fire size and the temperature are the main contributors to the fire throttling effect. The kinetic energy in the conservation of energy has been neglected on the basis that it is orders of magnitude smaller than the thermal energy [10]. The momentum source term due to buoyancy for the fire is neglected on the basis it is small compared to the momentum in the overall tunnel.

Separately, we are assuming that the friction losses at the location of the fire are negligible since the length of the fire location is small compared to the length of the tunnel, namely $F_f \approx 0$. These three assumptions simplify the calculation methods. We can obtain the pressure losses due to changes in temperature caused by the presence of the fire in two alternative ways.

First, combining equations (5.12) and (5.13), the pressure losses can be written as

$$\Delta p_f = p_2 - p_a = \rho_a u_a^2 \left(1 - \frac{u_2}{u_a}\right) \tag{5.15}$$

and substituting the ratio of velocities by the ratio of temperatures via equation (5.8), noting that $T_2 = T_s(0)$, we obtain the pressure losses as

$$\Delta p_f = \rho_a u_a^2 \left(1 - \frac{T_s(0)}{T_a}\right) \tag{5.16}$$

The temperature $T_s(0)$ can be calculated using the plume theory method as described in the later part of this same section. This version of the equation and calculation method is referred to as the *plume theory method*.

An alternative calculation of the pressure losses incorporates equations (5.12) and (5.13) into the energy equation (5.14), together with equation (5.8), to get

$$\frac{\dot{Q}}{A_T c_p} = \rho_a u_a^2 \frac{T_a}{u_a} - \rho_2 u_2^2 \frac{T_s(0)}{u_2} = \frac{T_a}{u_a}\left(\rho_a u_a^2 - \rho_2 u_2^2\right) = \frac{T_a}{u_a}\Delta p_f \tag{5.17}$$

The final expression of the pressure losses as a function of the HRR, $\dot{Q}$, is

$$\Delta p_f = -\frac{\dot{Q} u_a}{c_p A_T T_a} \tag{5.18}$$

This equation will be referred to as the *energy equation method*.

For the plume theory method in equation (5.16), the value of the temperature jump due to the fire, $\Delta T = T_s(0) - T_a$, can be calculated in two alternative ways.

The first one follows the suggestion by Ingason [46] to estimate the *average* temperature at the fire source as originating from the convective part of the HRR. From the control volume analysis carried out earlier, and replacing $\dot{Q}$ by its convective contribution $\dot{Q}_c$, namely

$$\Delta T = -\frac{\dot{Q}_c}{\dot{m}\, c_p} \tag{5.19}$$

where $\dot{m} = \rho_a u_a A_T$ is the mass flow of air in the tunnel. The convective HRR, $\dot{Q}_c$, is often assumed to be $\dot{Q}_c = 2/3\,\dot{Q}$ [46] which is the value that we will adopt here[1].

[1] The HRR ratio, $\dot{Q}_c/\dot{Q}$, is not constant in general. For instance, the Austrian code RVS 09.02.31 [59] recommends a value of the HRR ratio of 0.85 for 5 MW fires and of 0.75 for 30 to 50 MW fires, suggesting that the ratio should be, at least, a function of $\dot{Q}$ since as a larger fire causes higher temperature and more radiation.

The second one is obtained from a semi-empirical formula for the *maximum* temperature at the fire, $T_{\max}$, derived via the axisymmetric fire plume theory proposed by Li [58] as follows.

According to McCaffrey's fire plume theory [60], the maximum ceiling temperature can be assumed to be flame temperature when the fire is sufficiently large and impinges on the tunnel ceiling. In other words, maximum ceiling temperature should be a constant. This constant value was found to be approximately 1350°C based on several tunnel fire experiments [46].[2]

Relating interaction of fire plume and ventilation flow is a crucial consideration in application of fire plume theory. Therefore, a dimensionless ventilation velocity, v', was defined was defined [46] where both ventilation velocity, u_a, and characteristic plume velocity, w^*, can be correlated according to

$$v' = \frac{u_a}{w^*} \qquad\qquad w^* = \left(\frac{g\,\dot{Q}}{b_0\,\rho_a\,c_p\,T_a} \right)^{1/3} \tag{5.20}$$

where b_0 is the equivalent radius of the fire source, and g is the gravitational acceleration.

In this model, the maximum *ceiling* temperature increment, $\Delta T = T_{\max} - T_a$, is given in $°C$, by two separate formulas depending on the value of the non-dimensional ventilation velocity, v', namely

$$v' \leq 0.19 \quad \Delta T = 17.5 \frac{\dot{Q}^{2/3}}{H_{\text{eff}}^{5/3}} \tag{5.21}$$

$$v' > 0.19 \quad \Delta T = \begin{cases} \dfrac{\dot{Q}}{u_a\,b_0{}^{1/3}\,H_{\text{eff}}{}^{5/3}} & T_{\max} < 1350 \\ \\ 1350 - T_a & T_{\max} > 1350 \end{cases} \tag{5.22}$$

Finally we will obtain $T_s(0)$ using the convective part contribution of the HRR, i.e. substituting $\dot{Q}$ by $\dot{Q}_c$, in the calculation the value of $T_{\max}$ in the formulas above. Note that $T_{\max}$ is the value of the temperature at the ceiling, given that we use it here as a proxy for the smoke temperature

[2] We recognise the value of 1350°C is high, given it is the maximum temperature and that a lower representative value of 900°C has been reported in other fire plume and flames studies. We proceeded with 1350°C [61] as we will show later on this provides an upper bound value when compared to CFD models.

in all the perimeter of the tunnel section, we could expect pressure losses obtained with this parameter to be an upper bound. H_{eff} is the effective height of the tunnel.

5.5.4 Pressure losses in the hot smoke zone

The pressure losses, Δp_s, in the hot smoke zone ($2 \rightarrow 3$) are calculated by integrating the pressure gradient once the velocity is written in terms of the temperature, according to equation (5.9), as

$$\Delta p_s = \frac{1}{2}\frac{\rho_a u_a^2}{T_a D_h}\int_0^{L_s} \lambda_s T_s(x)\, dx \tag{5.23}$$

Here we will consider the friction coefficient in the hot smoke zone, λ_s, to be dependent on the velocity and thus on the temperature, therefore we will have $\lambda_s(x)$. This means that the pressure gradient in the hot smoke zone, $\frac{dp_s}{dx}$, will not be constant but a function of x of the form

$$\frac{dp_s}{dx} = \frac{dp_a}{dx}\frac{\lambda_s T_s}{\lambda_a T_a} = C_a \lambda_s T_s \tag{5.24}$$

where C_a is a constant. This expression, that relates the slopes of the pressure losses in the cold air and hot smoke zones of the tunnel, will be useful in analysing the results to be presented and discussed in Section 5.7. Note we have simplified this description by considering the deceleration of the flow and hence the recovery of static pressure to be negligible. For a longitudinally ventilated tunnel designed to minimise smoke back-layering, smoke is expected to be diluted (cooled) over a short distance from the fire, and therefore the temperature distance (cooling to the tunnel wall) over the remaining distance of the tunnel is not expected to be significant.

The adopted form of the temperature distribution, $T_s(x)$, in the hot smoke zone ($x \geq 0$) is defined as

$$T_s(x) = T_a + (T_s(0) - T_a)\,\mathcal{F}(x) \tag{5.25}$$

where the effect of the fire is assumed to be localised and represented as a sudden jump of temperature $\Delta T = T_s(0) - T_a$, and the decay of the temperature upstream of the fire is represented by the function $\mathcal{F}(x)$.

The temperature decay function $\mathcal{F}(x)$ could be modelled using heat transfer assumptions as suggested by Ingason [46] and calculated as the solution of the heat equation, i.e.

$$\mathcal{F}(x) = \exp\left(-\frac{h_t \pi D_h}{\dot{m} c_p} x\right) \tag{5.26}$$

which illustrates how the temperature decays as we move away from the fire in the direction of the flow. Note that h_t is the heat transfer coefficient (kW/m^{2}K), πD_h represents the wetted perimeter of the whole tunnel cross-section (m) and x is the distance from the fire along the tunnel (m).

Alternatively, one could adopt an empirical exponential form to fit an expression to experimental or field tunnel temperature data such as

$$\mathcal{F}(x) = 0.57 \exp\left(-0.13\frac{x}{H}\right) + 0.43 \exp\left(-0.021\frac{x}{H}\right) \tag{5.27}$$

where H is the height of the tunnel. This decay function has been generated from large-scale fire experiments [24] in tunnels, and the equation is documented in reference [46]. Equation 5.27 is used as a representative temperature profile that can be adopted in 1D calculations. This equation was chosen as it was derived from both the Runehamar and Memorial Tunnel fire tests and is found [46] to fit well to the experimental data. We acknowledge there may be more realistic or suitable temperature profiles that could be used. An investigation of the effect of different temperature profiles is outside the scope of this chapter, but incorporating them in future analyses should be straightforward.

5.6 Summary of the Pressure Losses Calculations in the TE1D Framework

This section collects the formulas, presented in the previous sections, for the calculation of the pressure losses along the tunnel due to the throttling effect. These calculations, which will be employed in the 1D code TE1D[3] framework, proceed as follows.

In the cold air zone of the tunnel upstream of the fire ($x < 0$) the pressure losses are given by

$$\Delta p_a = \frac{\lambda_a}{2D_h}\rho_a u_a^2(L_a - x) \tag{5.28}$$

At the location of the fire $x = 0$, the localised pressure loss is calculated in two alternative forms. The approach based on the energy equation writes the pressure losses across the fire in terms of the HRR, $\dot{Q}$, as

$$\Delta p_{f,e} = -\frac{\dot{Q}u_a}{A_T T_a c_p}, \tag{5.29}$$

Alternatively, we could use equation (5.16) with $T_s(0) = T_{\max}$ as per equation (5.25), leading to

$$\Delta p_{f,p} = \rho_a u_a^2 \left(1 - \frac{T_s(0)}{T_a}\right) \tag{5.16}$$

This formula refers to ceiling temperatures and this is somewhat inconsistent with the use of area-averaged quantities in the one-dimensional theory in which this framework is based. We will see however in Section 5.7 that the values obtained this way provide an upper limit boundary of Δp_f. In the following, we will refer to equation (5.16) as the plume theory method and to equation (5.29) as the energy equation method.

In the hot smoke zone, the pressure losses are calculated as

$$\Delta p_s = \frac{1}{2}\frac{\rho_a u_a^2}{T_a D_h}\int_0^x \lambda_s(t)T_s(t)\,dt \tag{5.30}$$

Given that the friction coefficient λ_s in the hot smoke zone will be in general a function of the location t through the velocity, that varies along the tunnel due to temperature changes as stated in equation (5.9), numerical integration will be required to evaluate the integral in equation (5.30).

[3]The TE1D (pronounced as TED) framework is available as a MATLAB code and can be obtained from the Zenodo repository [62].

An indicative sketch of the three contributions to the total pressure losses in the tunnel is depicted in figure 5.4.

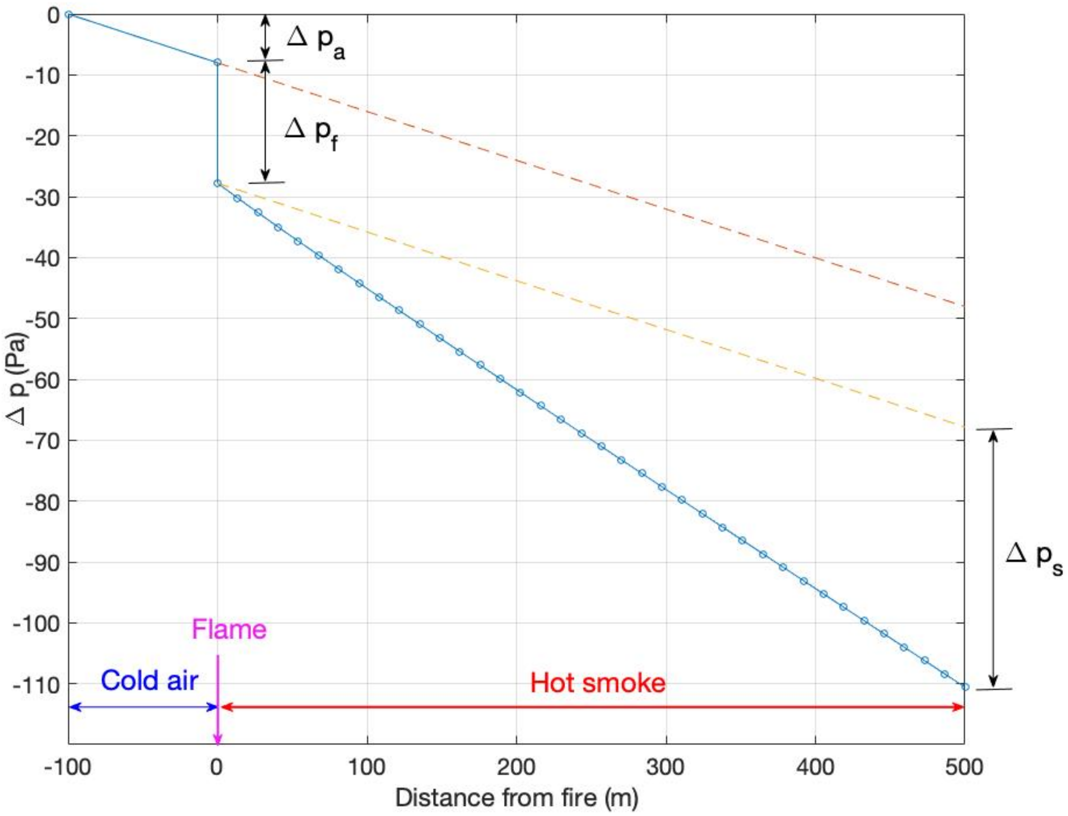


Figure 5.4: Schematic of the three contributions to the pressure losses along the tunnel: Δp_a (cold air), Δp_f (flame), and Δp_s (hot smoke). The pressure losses due to the fire throttling effect are $\Delta p_f + \Delta p_s$.

5.7 Verification of the TE1D Framework via Comparison to CFD Simulations

This section verifies the results from the 1D theory by comparison with CFD simulations obtained with OpenFOAM [10] and FDS 6.7.6 [14]. The objective of this comparison is to verify the consistency of CFD, and its suitability as a reference for the verification of the one-dimensional model, by considering two independently carried out simulations. FDS is the primary simulation tool used in this chapter. The OpenFOAM simulations have been carried out with FireFOAM, which is an application specifically designed for fire modelling, using a compressible LES model. More details on the FireFOAM simulations cited in this chapter can be found in the work by Riess [10].

We have used CFD simulations because there is a lack of suitable experimental or field data from real fires with measurements of both pressure loss and temperature distribution along the tunnel which would be required for validating the simulation and analysis of the throttling effect. Further, we have selected two sets of data, one from the FDS simulations by us, and another set of data carried out entirely independently by Riess [10] using FireFOAM. In addition to the data being from two different CFD models, this approach improves the independence and therefore the reliability of the comparison in particular because there are limited experimental and field data that provide the information we require.

Although some experimental and field data exist separately for either pressure or temperature[4], to the best of our knowledge, there is no data available for both in the same configuration. Tunnel fire experiments remain limited, and we take this opportunity to encourage development of more tunnel fire experiments especially considering the large number of new tunnel infrastructure to be built in the future [68].

5.7.1 FDS vs OpenFOAM Comparison

We have compared the FDS 6.7.6 results to those modelled independently by Riess [10] using OpenFOAM. The simulations model a 34 MW fire in a 600 m long rectangular tunnel at 10 (w) $\times$ 5.2 m (h) with a surface roughness of 2.5 mm. The fire is located at 100 m from the inflow portal, and the tunnel is longitudinally ventilated with an air velocity of 4 m/s. A uniform grid of 0.2 m is used noting the large fire sizes being modelled and therefore the grid is considered to be adequate. Adiabatic surface is used and is considered reasonable noting the simulations are run to reach steady state. The pressure profile in the FDS models are measured at fixed intervals as mean values across the cross section area of the tunnel.

[4] See for instance references [63, 64, 65, 66, 24, 67] to name but a few.

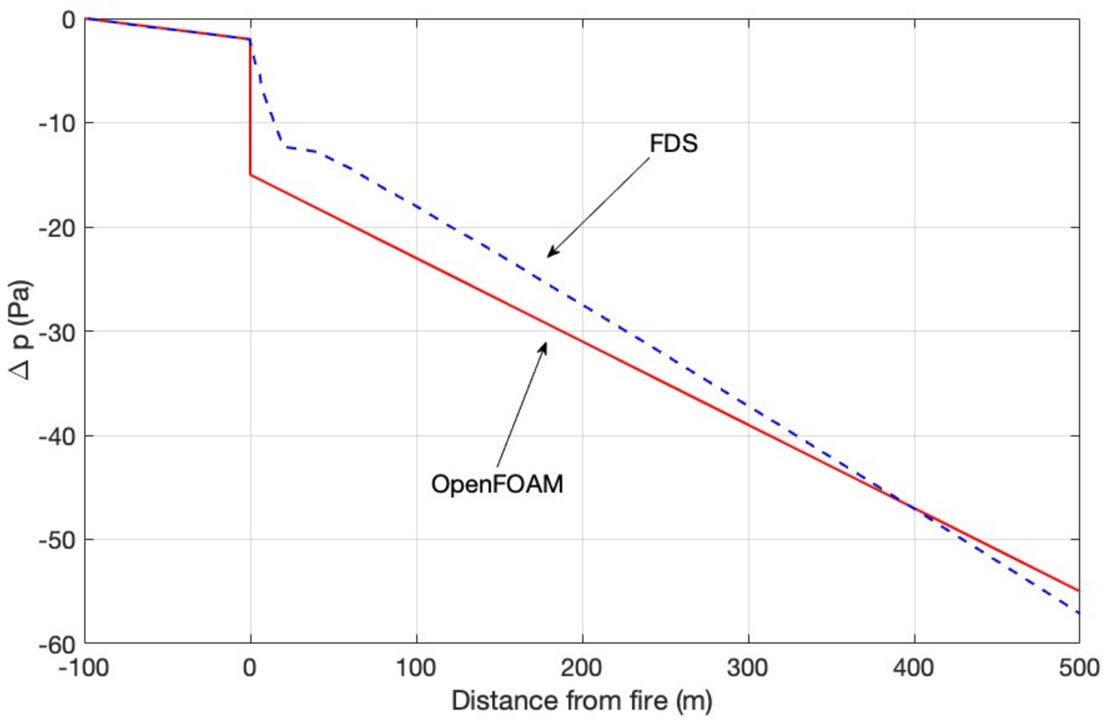


Figure 5.5: Comparison between the pressure losses calculated with FDS and OpenFOAM for a 34 MW fire.

The results in figure 5.5 show a reasonable agreement between the pressure losses calculated by OpenFOAM and FDS. The small discrepancies observed can be attributed to the difference in the method employed to measure the pressure drop. We have specifically chosen FDS 6.7.6 because its pressure solver has been modified to prevent the appearance of unphysical oscillations in the modelling of fires in long tunnels reported in previous versions [18, 17].

The subsequent three sections describe the evaluation of the terms Δp_a, Δp_f and Δp_s in the TE1D framework and verify them by comparing their values against those obtained in CFD simulations.

5.7.2 Calibration of the Term Δp_a

The value of the pressure losses, Δp_a, in the cold air zone upstream of the fire, depends on the surface roughness coefficient, ε, in the friction factor given by equation (5.7). Here we choose the value of ε that matches the pressure gradient in the cold air zone calculated by the TE1D model to the pressure gradient in both CFD simulations. Figure 5.5 shows that both CFD models produce the same pressure gradient in the cold air zone.

5.7.3 Verification of the Term Δp_f

This section compares the CFD results with those obtained using the energy and plume equations in the TE1D framework, i.e. equations (5.16) and (5.29) respectively, and with the semi-empirical formulas:

$$\Delta p_f = C_1 \frac{\dot{Q}_c}{u \cdot D_h^2} \tag{5.31}$$

$$\Delta p_f = C_2 \frac{\dot{Q}_c^{0.8} \cdot u^{1.5}}{D_h^{1.5}} \tag{5.32}$$

where C_1 and C_2 are empirical constants, $\dot{Q}_c$ is the convective component of the fire HRR, u is the airflow velocity, and D_h is the hydraulic diameter of the tunnel cross section.

Formulas (5.31–5.32) have been proposed by CETU [9] and Dutriueue and Jacques [8] respectively, and their empirical constants C_1 and C_2 have values $C_1 = 9 \times 10^{-5}$ and $C_2 = 41.5 \times 10^{-5}$ $\mathrm{s^{1.9} kg^{0.2}/m^{2.6}}$.

Figure 5.6 shows a comparison of the values of Δp_f for different fire sizes calculated by TE1D, the semi-empirical formulas (5.31–5.32), FDS and OpenFOAM [10]. The main differences are the fire size and the ventilation velocities of the rectangular and arched tunnel profiles.

By considering the plume method and the energy method as an upper and lower bound, the comparison shows the TE1D prediction is within this boundary compared to CFD models. In practical applications where there can be uncertainties, in particular early on the project where 1D method is most likely used, we recommend if TE1D is used to predict Δp_f, that TE1D plume theory is used. This is because the over prediction of the pressure drop will provide a level of safety buffer, and as per figure 5.6, TE1D energy equation under predicted the pressure drop when compared to the OpenFOAM values [10].

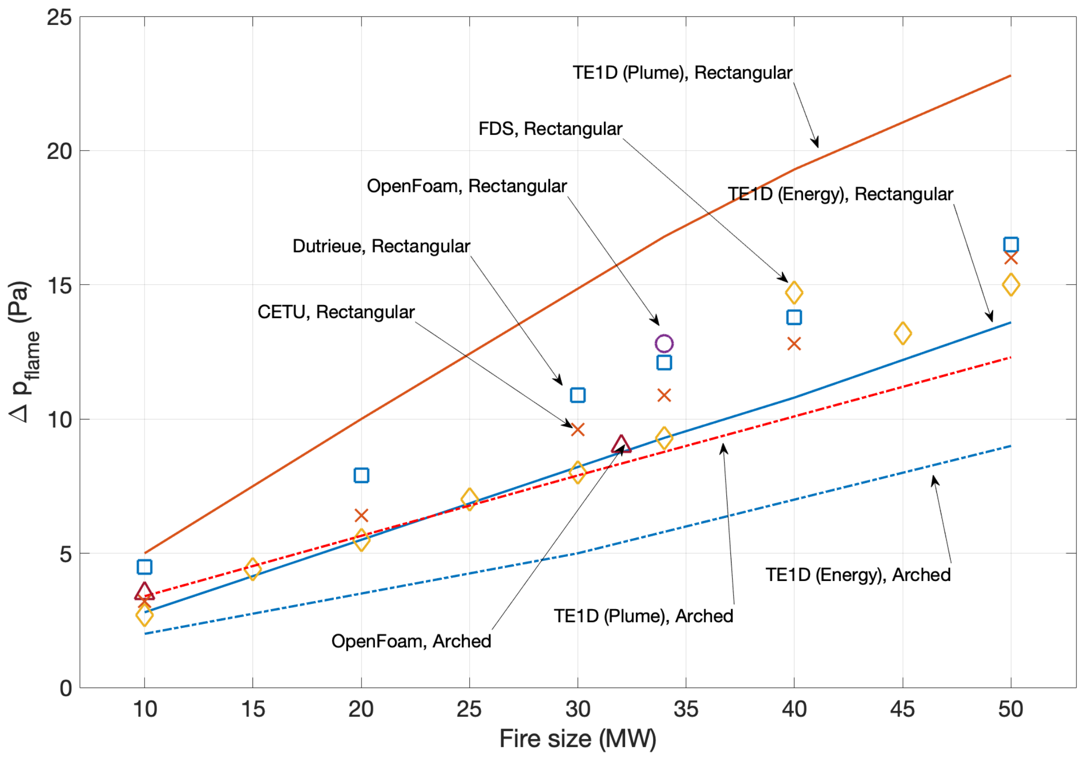


Figure 5.6: Comparison of the pressure drop Δp_f calculated using the TE1D energy and plume theory methods with values calculated by FDS and OpenFOAM and by using two semi-empirical formulas proposed in the literature [8, 9]. Arched tunnel sections included to provide additional comparisons.

5.7.4 Verification of the Term Δp_s

The simulations of the pressure losses in the hot smoke zone show that the pressure gradient is approximately constant for all the fire sizes considered. Therefore, to verify the 1D model we have compared the values of the pressure gradient obtained by TE1D with those evaluated using FDS and OpenFOAM [10] for different fire sizes. Figure 5.7 shows the relative error of the value of the pressure gradient in TE1D with respect to the CFD value, which is used as reference, for a HRR range. Note that the equations (5.31–5.32) proposed by CETU [9] and Dutrieue and Jacques [8] respectively only account for localised pressure losses at the fire and therefore have not been used for comparison here.

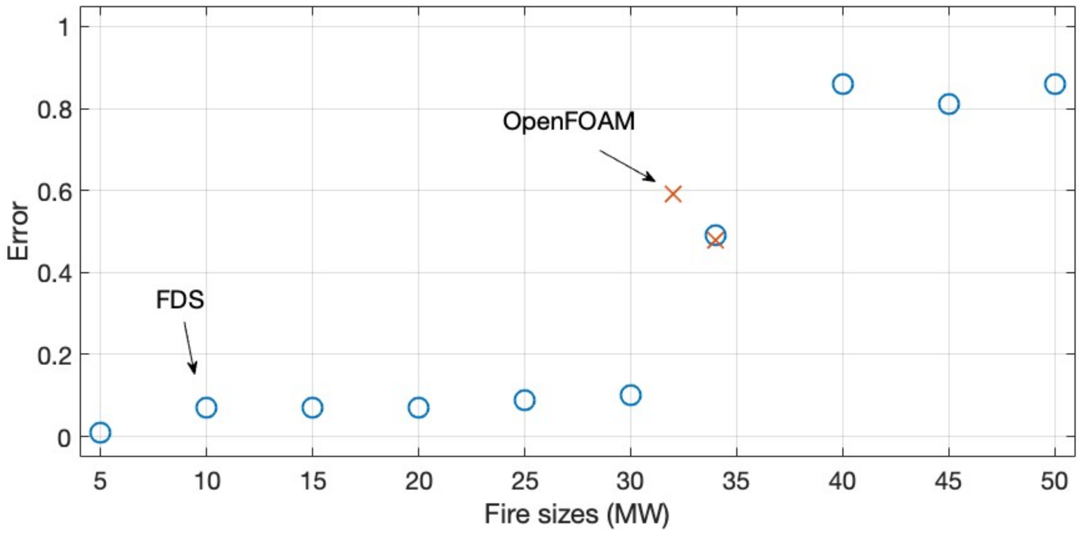


Figure 5.7: Comparison of error of TE1D to FDS and OpenFOAM [10]. The error is calculated as $|1 - \epsilon|$ where ϵ is the ratio of the slopes calculated using TE1D and CFD.

The comparison in figure 5.7 shows for a fire size under 30 MW, the error is tolerable when viewed from a preliminary design for practical application perspective. The results show beyond 30 MW, the error increases significantly. Keeping in mind the only variable here is the fire size with the velocity and tunnel geometry remain consistent, we postulate temperature alone when it comes to a larger fire size is not the critical contributor to pressure loss in the hot smoke zone.

To confirm this, using the FDS model we obtained the average of the temperature distribution in the hot smoke zone downstream of a 34 MW fire, and derived a fitted temperature curve given by $T_s(x) = T_a + 107\, e^{(-0.019x)}$. The corresponding pressure drop Δp_s is calculated using this fitted temperature distribution and it is shown in figure 5.8.

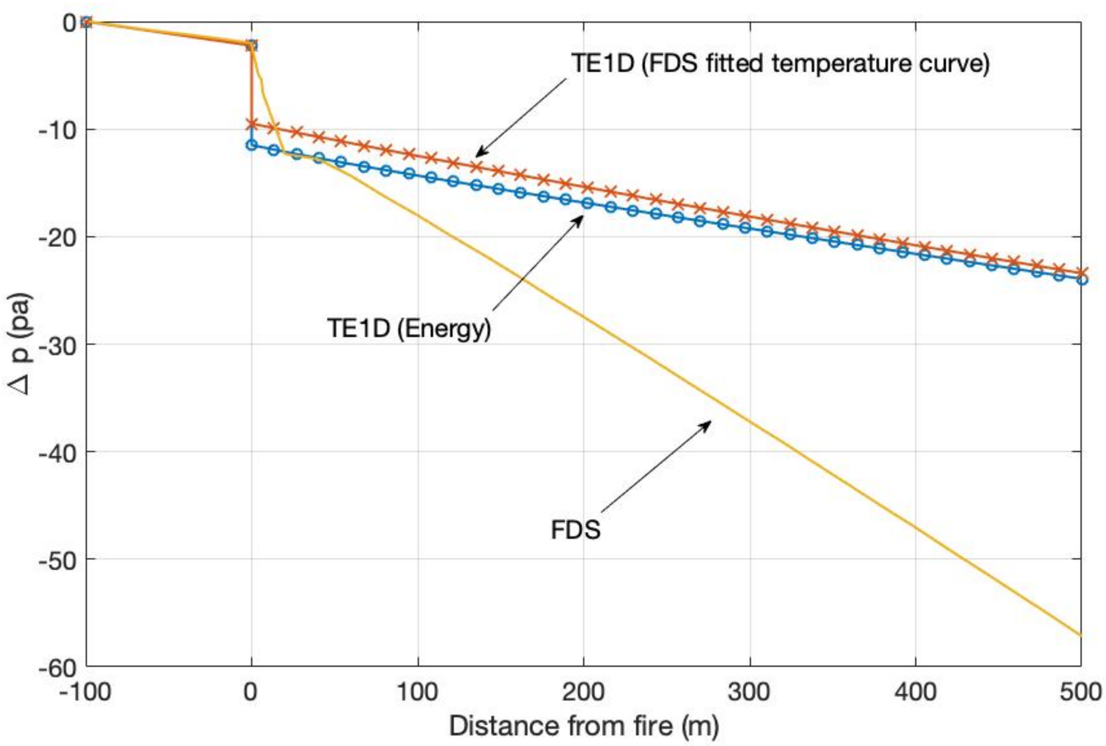


Figure 5.8: Comparison of the slope between TE1D derived temperature curve, and FDS fitted temperature curve to FDS model result for a 34 MW fire.

The results in figure 5.8 show with the fitted temperature curve, the slope is marginally steeper, i.e. closer to the steeper FDS slope, but the difference is negligible. This supports our suggestion for a larger fire the temperature is not the main contributor to the pressure loss in the Δp_s region.

Recall equation (5.24) that relates to the slopes of the pressure losses in the cold air and hot smoke zones of the tunnel. Figure 5.9 shows the slopes for different fire sizes. The results show the effect of the additional pressure losses due to the increased temperature reduces materially at about 150 m away from the fire, which does not explain the significant difference in the slope shown in figure 5.7 and figure 5.8 for the bigger fires.

This is further reinforced by the plot in figure 5.10. This figure that shows that if temperature were the main contributor, we would expect the λT curve to show a much larger difference between the TE1D temperature and the FDS fitted temperature curve, as opposed to a negligible difference in the region of 5 to 6 observed here.

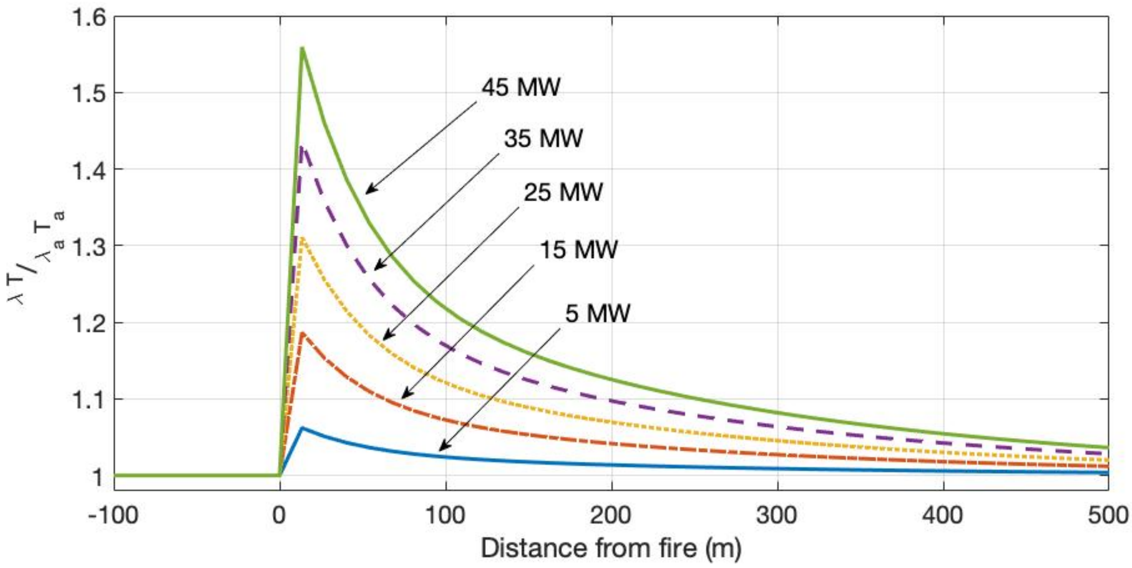


Figure 5.9: Comparison of $\frac{\lambda T}{\lambda_a T_a}$ for 5, 15, 25, 35 and 45 MW fires.

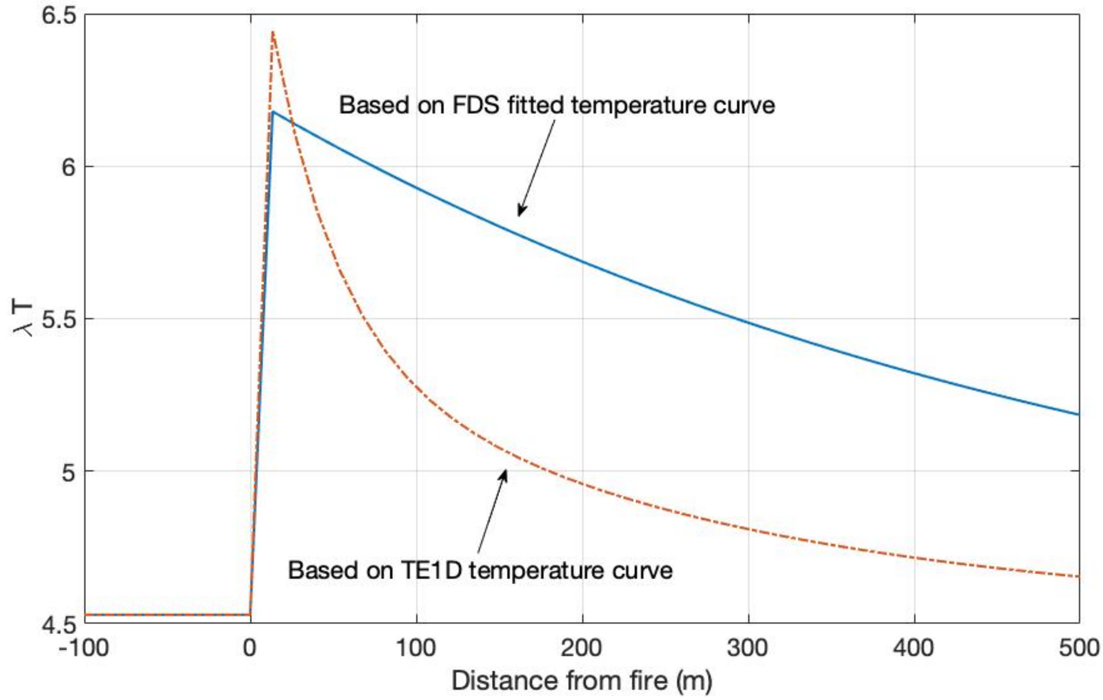


Figure 5.10: Comparison of λT using TE1D temperature curve and FDS fitted temperature curve for a 34 MW fire.

The analysis with TE1D so far shows the increased friction due to changes in velocity (increased in temperature) is sufficient to capture the effect of the fire for smaller fire sizes. However for larger fire sizes beyond 30 MW, the increase in temperature alone does not fully account for the pressure loss due to the fire throttling effect in the hot smoke region, Δp_s.

As suggested by Riess [10], temperature stratification and the resultant shear stress to the tunnel walls appears to play a significant contribution to pressure loss in the Δp_s region. Figure 5.11 shows a comparison of the 2D velocity profile and temperature stratification for a small and a large fire, where the large fire can be seen to have much greater smoke stratification at the ceiling level.

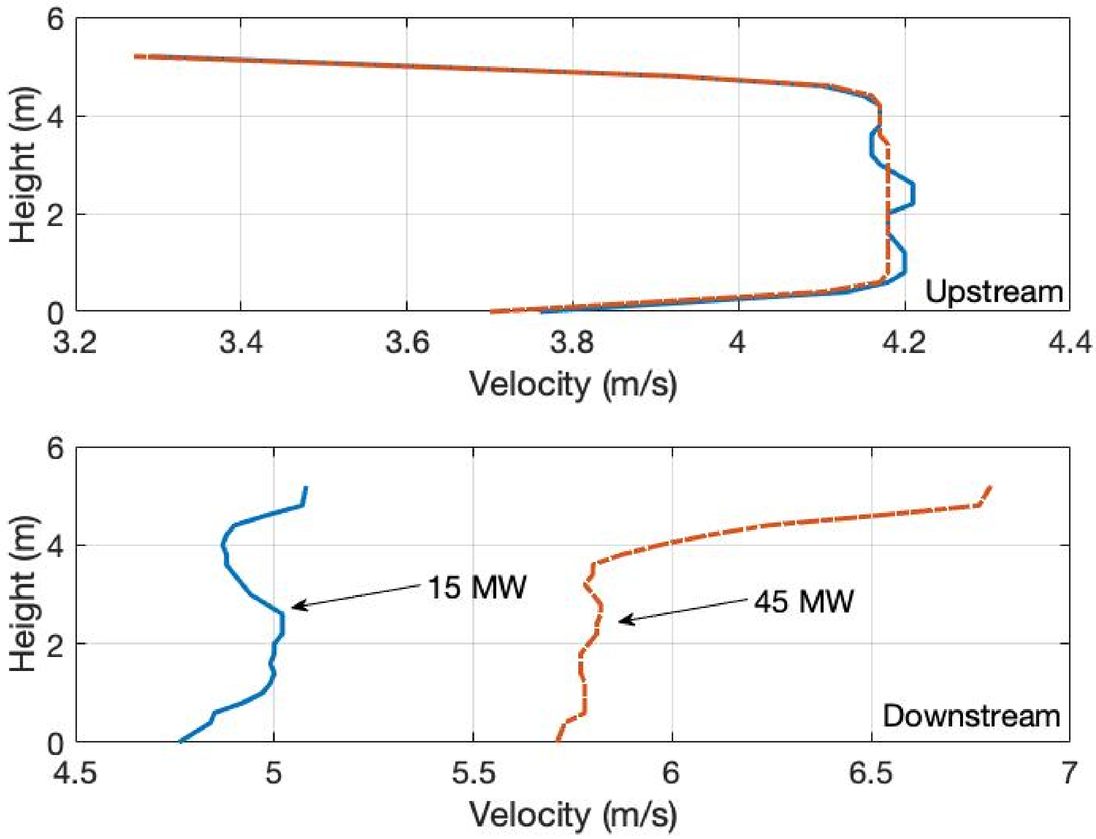


Figure 5.11: Comparison of the temperature stratification downstream of the fire for a small and a large fire. Minor fluctuations in the upstream velocity profile for 15 MW fire are due to the data collection.

Based on the above, this means for a 1D model and a large fire, e.g. beyond 30 MW, the increased temperature alone downstream of the fire is not the main contributor to the pressure loss. Considering the results of the comparison between 1D and CFD models in this chapter, we conclude that a 1D model, when there is a sufficiently large fire where significant stratification is expected to occur, cannot capture the 3D effect of a fire, which is the bigger contributor to the pressure loss. This means when dealing with scenarios where large fires or where significant stratification are expected, additional corrections are required for a 1D model for the accurate prediction of pressure loss downstream of the fire, or Δp_s. One example of a correction is proposed by Riess [10]. Alternatively, CFD models should be used in these scenarios.

Conversely, we have now identified a threshold value where a 1D model, e.g. TE1D, holds up reasonably well for fire sizes up to 30 MW. We point out the 30 MW is related to the limitations of TE1D in predicting fire throttling, and that this is not a universal fire size that determines the occurrence of stratification that results in pressure losses. We acknowledge the cases tested in this research may not be universal, and that a parametric study considering the tunnel geometry, ventilation velocity and other variables that impact the stratification beyond fire sizes should be considered.

The analysis in this chapter is based on a tunnel with a homogeneous profile unlike a wider tunnel such as a rectangular tunnel commonly seen in a cut and cover or mined tunnel section, as opposed to a bored tunnel (circular profile). In the instance where a tunnel has a wider or non-homogeneous profile, it is possible a fire size larger than 30 MW is required to trigger the non-neglibible pressure losses of fire throttling. This is because for a wider tunnel such as a three car lanes tunnel, the effect from the smoke and heat of the fire may still be localised to approximately one lane, rather than over the entire cross section of the tunnel. This means the flow downstream of the fire may not reach a state where the bulk flow consideration applies.

Conversely for a narrower or smaller tunnel, a fire smaller than 30 MW may trigger the non-negligible pressure losses of fire throttling. This is because with a smaller tunnel cross section, a smaller fire size can cause the flow downstream of the fire to reach a state where bulk flow consideration applies.

That said, from a practical application perspective, especially for the early stages of a project where speed of computation matters more and that designers are only interested in an approximate tunnel ventilation requirement, 1D models such as TE1D can be considered to estimate the impact of fire throttling effect, in particular for smaller fire sizes, e.g. 5 to 15 MW that reflects a typical passenger car or metro train design fire sizes.

5.8 Conclusions

In this research, we have firstly amalgamated the previously fragmented understanding of fire throttling. We do this by using a one dimensional model. The model, implemented in the code TE1D, analyses the contributions to the pressure losses caused by the fire by splitting the tunnel length downstream from the fire into flame and hot smoke zones, and using the temperature distribution along the tunnel to account for the effect of the fire on pressure losses. The 1D model used here is based on a longitudinally ventilated tunnel, with adequate velocity such that back-layering can be minimised.

With the 1D model established, we then compared the prediction using this model to CFD models for varying fire sizes from 5 MW to 50 MW. The comparison shows the 1D model holds up well within the predictions between TE1D Plume and Energy methods in predicting the localised pressure drop at the fire location across the varying fire sizes. For the region downstream of the fire, the comparison shows 1D model predicts reasonably at approximately 10% error for fire sizes up to 30 MW, and subsequently under predicts the pressure losses for large fire sizes beyond 30 MW with an error greater than 50%.

By using a temperature curve derived from the CFD model, we have shown for larger fire sizes, the increase in temperature alone is not the main contributor for the pressure losses downstream of the fire. We believe the 3D effect of temperature stratification and the resultant shear stress effect on the tunnel wall could be a main contributor to the pressure loss. Additional correction is needed to the 1D model to account for the full impact of the fire throttling effect for a larger fire beyond 30 MW where significant stratification is expected to occur.

For smaller fire sizes, the 1D model can be used in particular for earlier stages of design where engineers needed to prioritise approximating the fan thrust needed to overcome the losses expected in a tunnel.

We acknowledge the calculation methods here rely heavily on CFD models. This is long known where the lack of validated CFD models remain a challenge that has faced the tunnel fire safety community for a long time [18]. Unlike a safety critical but simple problem, the calculation of pressure losses is safety critical in the design of a tunnel ventilation system, and the presence of the fire throttling effect complicates the calculations.

With a safety critical and complex problem, this issue requires significantly more investment including tunnel fire tests to better understand the fire throttling effect, and to provide better validation benchmark. We strongly advocate that the tunnel fire safety community needs to continue promoting the importance of fire tests to support better 1D and 3D model validation, in particular as these computational methods are now the primary tools for tunnel fire safety design.

This research has gathered and analysed some of the ideas and methods proposed in the available literature to model the throttling effect with a view to gain a better understanding of the phenomenon and the limitations of these methods, and to develop a one-dimensional framework (our TE1D code) as a starting point for engineers and researchers to build upon when considering the fire throttling effect.

We conclude with a quote from George Box [69]: *All models are wrong, but some are useful.* This is a reminder on the importance to understand the limitations of models, including 1D models prior to using them. Furthermore, it is our belief that improved, more accurate, models would be developed and validated if full scale tunnel fire tests relevant to the fire throttling effect were to be performed and the acquired data made available to researchers and engineers in the fire ventilation community.

Chapter 6

Discussions

6.1 Outcome of this Thesis

We set out in this thesis to understand the state of the art in engineering simulation based on 1D analytical mode, and CFD, in particular FDS. *'Are engineering simulations the right tools for tunnel fire modelling?'* The short answer, is yes. The longer answer, is summarised below.

The status quo in the industry is that these tools are now widely used in engineering design, and there is in fact an expectation for them to be deployed from small to large scale projects. This thesis has reiterated three main considerations:

Experimental Validation and Repeatability

Through demonstrating the variability associated with carbon monoxide modelling, and the variability in the experiments, we show the importance of understanding the physical reality constraints numerical methods. If we cannot understand the **experimental validation and repeatability** to the physical reality, this uncertainty cascades into the model which is no doubt a simplified version of the physical reality.

We learnt that for a common material in research, like wood, the CO production of the material still has significant variability in repeated experiments. The variability currently is not explicit when engineers select the CO yield to be used in simulation. We showed that for the same material, various researches [52] [30] [5] publish different CO yield depending on the combustion, i.e. oxygen rich or ventilation controlled, and this could have different CO production. Given the complexities associated with a common material like wood, there have been researches that show treated timber, e.g. those used in construction, has different level of CO production compared to untreated timber. We are only scratching at the surface of complexities, as we have not dived into the world of composite materials that are commonly used in construction today.

Engineers are strongly advised to consider a range of CO yield for the same material, and to consider the combustion conditions where the CO yield is based on. A sensitivity study should be carried out for the range, and if a range of value is unavailable, engineers are advised to consider the use of a safety factor. Based on our research here, we consider a safety factor of 2 should be considered for the CO yield selected. Like all engineering design, it is up to the engineers' judgement to determine the appropriate CO yield to use, and its impact on the practicality and safety of the design.

Limitations of the Solver

We demonstrate the importance of validation through the investigation of the numerical oscillation observed when modelling a large fire in a longitudinally ventilated tunnel. The phenomenon of oscillation in this scenario has never been widely reported in the literature. It encouraged us to better understand this, and as a result our work contributed in the improvement of FDS. Our work here demonstrates the importance in understanding the **limitations of the solver**.

For tunnel fire modelling, smaller fires for example those under 30 MW, would not trigger significant oscillations. Translating this to engineering designs, most fire sizes used in the design of metro trains would be below this, with the exception of inter-city trains, or trains with a plush first class carriage. This fire size is expected to be exceeded in road tunnels where trucks and medium or heavy goods vehicles are present. However, even when modelling a small tunnel fire in FDS, the PRECONDITIONER function should be used instead of one cell sized vents, which we termed micro-vents to differentiate from a 'real' vent like an exhaust vent. The use of micro-vents to attenuate the oscillation is an unrealistic physical solution to a numerical issue as these micro-vents does not reflect the real characteristics of a long tunnel. We are reminded by Gresho *et al* [40] to not ignore the wiggles as they are telling you something. The convergence of the pressure solver in the FDS solver should be monitored, and increased if the solver consistently runs at the maximum iterations.

Both 1D and CFD Simulations are Useful

As we better understand the capabilities of CFD and FDS, we wanted to use engineering simulations to better understand fire throttling. We showed **1D and CFD simulations are both useful**. There remains an important place for 1D model in engineering design, and concurrently CFD remains critical as 1D model has limitations which we needed to understand. On top of this, our work here contributed in amalgamating the currently fragmented understanding of fire throttling.

Using the simple TE1D model, we showed that fire throttling occurs not only locally at the location of the fire, but also downstream of the fire. We showed fire throttling can be reasonably predicted using a 1D model, TE1D our model in this case, for fire sizes up to 30 MW. Coincidentally this is also the point at which oscillation becomes significant. Beyond 30 MW is the point that the 3D effect of temperature stratifications is the main contributor to pressure losses downstream of the fire. Through comparisons between TE1D and CFD, we showed for a large fire that a 1D model, unless specifically corrected, is unable to account for this 3D effect. CFD however, based on two independent OpenFOAM and FDS simulations, is able to model and quantify fire throttling.

By amalgamating our findings in both 1D and CFD simulations, we posit that the contributions to fire throttling occurs at two locations, locally at the fire's location, and downstream of the fire. The 3D effect downstream of the fire becomes the main contributor to pressure losses for fire sizes beyond 30 MW, and this effect cannot be captured in a 1D model unless additional considerations are incorporated. To the best of our knowledge, we were unable to identify a peer reviewed publication that describes how this effect can be considered in a 1D model. We note there is some discussion [25] suggesting this effect can be considered but we were unable to locate a quantitative description of this. The 3D effect from large fires, i.e. those beyond 30

MW, can result in greater pressure losses, and this effect needs to be considered when designing tunnel jet fans.

6.2 Real World Application

We can illustrate the issues discussed in this thesis using a simple example based on the Dartford Tunnel (see Chapter 4). Recall this is a 1.5 km long tunnel with a cross section of 40.96 m^2 and it has a ventilation velocity of 4 to 5 m/s. For this road tunnel, and assuming a fire on a truck carrying goods, this could result in a fire size of 55 MW or above.

Regarding a fire size of 55 MW, in designing a tunnel ventilation system, designers consider the possible peak heat release rate of a fire without factoring in the fire suppression. This is an implicit approach of multiple layers of safety, i.e. if the fire suppression system worked as intended, any small vehicle fire would be quickly put out before it turns into a bigger fire. The design of the ventilation system is an added layer of safety in case the fire can only be suppressed, but not extinguished.

Back to the 55 MW fire, recall figure 4.2 where the modelling shows an oscillation between 160 kg/s and 240 kg/s, whereas the cold flow (no fire) model has a constant mass flow of 240 kg/s. Noting that there are 14 pairs of fans, with each providing a volumetric flow rate of 8.9 m^3/s [70], and assuming a simple mass flow equation, we establish a fluctuation between 160 to 240 kg/s means a difference of approximately 4 pairs of fans. These uncertainties could lead designers to question the need to provide additional jet fans beyond the 14 pairs to achieve 240 kg/s in a fire, or if this number is already adequate.

This uncertainty is further compounded if an engineer is looking at the dilution rate required to reduce the concentration of carbon monoxide. The engineer is faced with another uncertainty on top of that associated with modelling of carbon monoxide in the form of a very large variability in the carbon monoxide yield.

The work in Chapters 3 and 4 proposed a way to alleviate these uncertainties. The engineer can now use a revised version of FDS whereby the numerical oscillation is negligible, and understand how carbon monoxide yield could be modelled for a steady state fire.

Using the same example, the 1D calculations using TE1D show a pressure drop of 25 Pa at the fire, and a pressure loss of 60 Pa downstream of the fire due to fire throttling. Recall this is the additional pressure loss due to the fire and it is equivalent to approximately 3485 N of resistance for the 41 m^2 cross section.

Although the jet fan product data for Dartford Tunnel is unavailable [70], using a similar product [71] and working backwards using the known velocity of the jet fans at 34 m/s [72], we estimate the fan in Dartford Tunnel is a 1 m diameter fan with a thrust of 1043 N. Note a 1 m diameter is typical in a road tunnel.

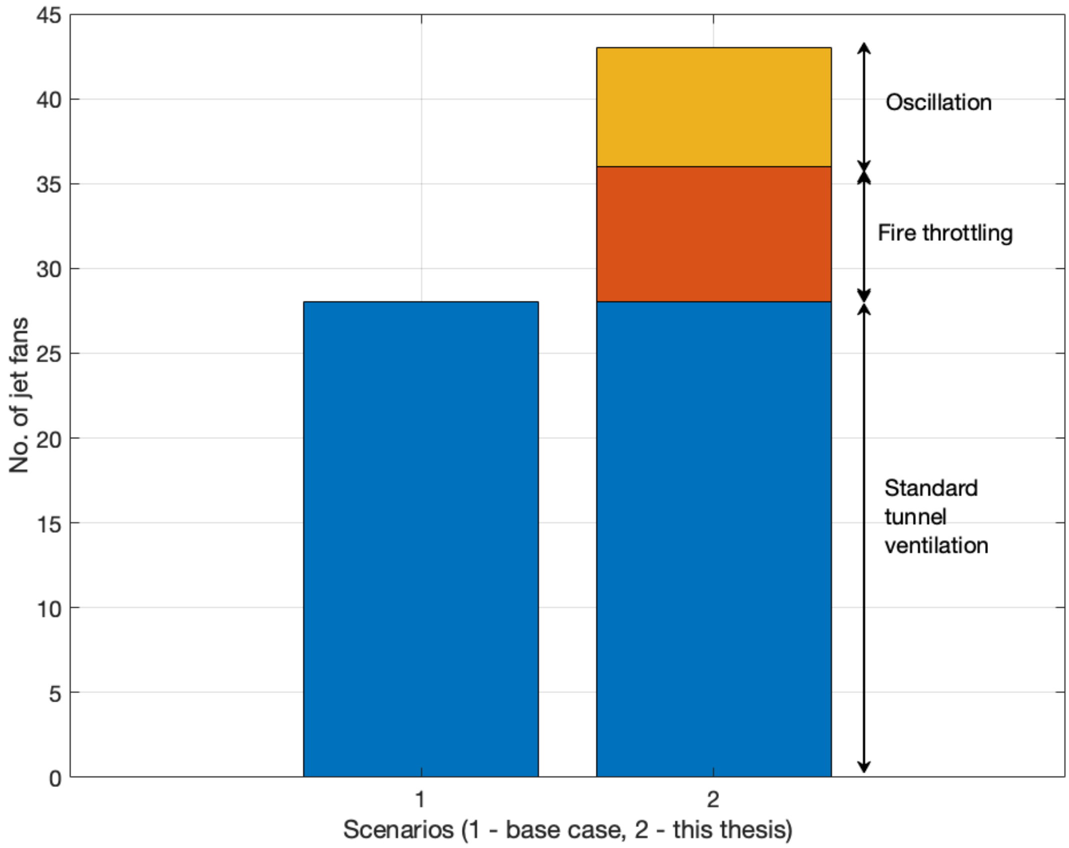


Figure 6.1: Comparison between the number of jet fans between base case Scenario 1 without fire throttling or oscillation, and Scenario 2 with fire throttling and oscillation.

Applying a typical installation factor of 0.5 to represent the various efficiency losses of the fans due to the proximity to the tunnel soffit and other installation caused losses, we get a useable fan thrust of 521 N per fan. The additional 3485 N of thrust required to overcome fire throttling represents an additional 7 jet fans (or 4 pairs, rounded up because in the real design, there needs to be a redundancy in the provision of the fans). This is illustrated in figure 6.1.

If uncertainty associated with the oscillation is also accounted for, we are looking at a total increase of 8 pairs of jet fans. This uncertainty not only has a significant impact on the design of the tunnel ventilation system, but also on the design of the wider tunnel, as the fans would require an additional 640 kW (40 kW per fan) of power, which significantly increases the tunnel substation maximum demand requirements. The maximum demand in a tunnel is usually sized with a 20% spare capacity on top of the maximum usage, and these additional fans could tip the calculations to the point a larger capacity substation would be needed. On top of this, additional pits and under road conduits would be required to accommodate the power cables required for the jet fans. Any design changes to the main infrastructure such as the substations, and to the tunnel structures for the additional pits and conduits represent a significant cost increase that could break the financing of a project.

We acknowledge the above is an example using a simple engineering rule of thumb only. However, it illustrates the challenges associated with not considering the issues outlined above.

6.3 Future Work

Computational methods critically rely on real world experiments to serve as validation cases, but often these experiments are not designed with CFD validation in mind. We strongly believe there should be greater collaboration between the computational and the experimental communities.

For example, the developers for FDS have mentioned they would love to have more validation cases (experiments) to be added to their repository, and our suggestion to the tunnel fire safety community is to reach out to these developers (not just FDS) and computational communities as you are planning your experiments.

Another important issue is the repeatability of experiments. This topic is not often considered, and, as we have shown in Chapter 3, the experimental challenges experienced in repeating experiments can inform engineers that some simulations, for instance if carbon monoxide yield, are inherently difficult to predict in the real world, and this carries into simulations.

There are three future research directions that we propose for the predictions of carbon monoxide explored in Chapter 3:

1. To explore the development of uncertainty models that can provide a variance of the CO yield that can be expected of a specific material. This variance can help engineers understand the possible threshold of CO yield that should be considered for a material and would allow the engineers to decide the suitable CO yield that can be used.
2. In future experiments and design guidance updates, the combustion conditions in which a CO yield has been derived from should be explicitly noted so that engineers can consider its implication when assessing the suitability of a CO yield.
3. To explore the repeatability of experiments or simulations introducing other materials commonly considered in practical application, such as polyurethane and compare the repeatability to a priori study using simulations. Although there are currently numerous modern research into fire resistance of timber or concrete, there remains limited number of researches associated with understanding material properties and species productions in combustion.

For Chapter 4, where we explored the oscillations that appear when modelling a large fire in a longitudinally ventilated tunnel using FDS, we mistook our initial observation of the oscillation to be a pulsation effect observed in experiments [38]. It would be beneficial to the fire safety community and engineers to better understand this phenomenon in simulations. To our knowledge, there has so far only been one attempt by Riess [10] to shed some light to this problem.

For Chapter 5, where we attempted to better understand and describe fire throttling using a one-dimensional model, there are several suggestions for future work:

1. To undertake a parametric study to establish the impact of fire throttling in tunnels with varying geometry (length, width and height) and varying profile including round TBM (tunnel boring machine), cut and cover tunnels with rectangular profile and mined tunnels with oval or elliptical shapes. The parametric study should seek to establish the conditions for fire throttling to become a material issue in design, and whether this would occur in a wider tunnel, where the fire only occupies one lane of a three lane tunnel for example, or in a taller tunnel, for example in the cavernous area of a tunnel. This study should result in a set of non-dimensionalised parameters for the primary factors, e.g. width, length of height of the tunnel, and a fire size where fire throttling becomes non-negligible.

2. To expand TE1D to include / modelling smoke backlayering. One idea to achieve this is to first include a check variable to test if based on a certain velocity where backlayering could occur. If backlayering could occur, an additional pressure drop zone could be added between the cold zone and the flame zone to account for the presence of backlayering. There are equations available based on current research to estimate the extent of backlayering.

3. To carry out additional simulations to understand the relationship between the additional pressure loss downstream of the fire caused by the 3D effects as a result of a large fire. If a correlation can be established, a semi-empirical approach could be introduced to TE1D to improve its accuracy for fire sizes beyond 30 MW.

4. To develop a 1D two-phase flow model to explore if such model could simulate stratification and account for the 3D effects of a large fire and the additional pressure losses downstream of the fire.

Chapter 7

Conclusion

In this research, it was shown that CFD and FDS are powerful tools for tunnel fire modelling, however, for them to be the right tools, this requires care from the engineers to understand the limitations of these tools. When used correctly, CFD and FDS have helped us to better understand fire phenomena such as the fire throttling.

Throughout this thesis, there is continuously the same challenge. The tunnel fire safety community is a subset of a small niche fire safety community. The CFD model validation is an important aspect of CFD code development, and in particular for FDS that is based on LES where engineers need to understand the length scales to filter (mesh size dependent). FDS and all CFD models are simplified versions of the physical world, and it is only through fire experiments and large scale fire tests that these CFD models can be validated. There remains a very small number of suitable fire experiment for code validation. As a case in point the FDS community over a decade has only collected over 40 experimental cases to validate FDS, and less than 10 % of these are based on a tunnel fire experiment.

It is therefore critical for the community to have access to more relevant fire experiments that can form a basis of validation for CFD models, and assist engineers and researchers to better understand the limitations of these CFD models.

CFD models, and engineering design will continue to grow in complexity resulting, in some instances, in weeks of effort to prepare a CFD model and several more for the simulations and post processing. There are time consequences in engineering design to re-run a model because more often than not, there is not enough time buffer in a design program to revisit CFD models.

This thesis is concluded with a reminder for engineers to be ‘intelligent users’ of the computational tools, whether these are 1D or 3D engineering simulations. It is only by better understanding and validating the capabilities of these tools that would enable engineers, as tunnel ventilation engineers, to design safer and more efficient infrastructure. Finally, this thesis is closed with a maxim from Ove Arup [73], the brilliant engineer behind Sydney Opera House, ‘If you do not know the order of magnitude of the answer, do not use the computer.’

Bibliography

[1] Kuan Lin. 1D assessment of fire throttling effect in tunnel. Final year project report, Department of Aeronautics, Imperial College London, 2019.

[2] Transport for NSW. *TP04 Road Tunnel Ventilation Systems*. Transport for New South Wales, 2014.

[3] C. D. Ang, G. Rein, J. Peiró, 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*, 52:119–126, 2016.

[4] C. D. Ang, G. Martin, and D. Weinert. Unidrectional ventilation in rail tunnels. In *Australasian Tunnelling Conference 2020+1*. Australian Tunnelling Society, Australian Tunnelling Society, 2021.

[5] Morgan J. Hurley, editor. *SFPE Handbook of Fire Protection Engineering*. Springer, fifth edition, 2016.

[6] S. Hostikka, E. Veikkanen, T. Hakkarainen, and T. Kajolina. Experimental investigation of human tenability and sprinkler protection in hospital room fires. *Fire and Materials*, 45:823–832, 2020.

[7] T. Melcher, R. Zinke, M. Trott, and U. Krause. Experimental investigation on the repeatability of real scale fire tests. *Fire Safety Journal*, 82:101–114, 2016.

[8] R. Dutrieue and E. Jacques. Pressure loss caused by fire in a tunnel. In *12th International Symposium on Aerodynamics and Ventilation of Vehicle Tunnels*, pages 77–84, Portoroz, Slovenia, 11–13 July 2006. BHR Group.

[9] Centre d'Études des Tunnels. *Les dossiers pilotes du CETU – Ventilation.* CETU, 2003.

[10] I. Riess. Aerodynamic resistance of fires in tunnels. Technical report, Riess Ingenieur-GmbH, December 2020.

[11] N. Casey. Fire incident data for australian road tunnels. *Fire Safety Journal*, 111(102909), January 2020.

[12] Transport for NSW. Transport for NSW Traffic Volume Viewer, July 2022. https://roads-waterways.transport.nsw.gov.au/about/corporate-publications/statistics/traffic-volumes/aadt-map/index.html/?z=15lat=-33.859593693114704lon=151.21827297789642st=3yr=2018.

[13] A. Beard and R. Carvel. *Handbook of Tunnel Fire Safety.* ICE Publishing, Second edition, 2012.

[14] K. McGrattan, S. Hostikka, R. McDermott, J. Floyd, C. Weinschenk, and K. Overholt. *Fire Dynamics Simulator User's Guide.* NIST and VTT Technical Research Centre of Finland, sixth edition.

[15] K. McGrattan, S. Hostikka, R. McDermott, and M. Vanella J. Floyd. *Fire Dynamics Simulator Technical Reference Guide - Validation.* NIST and VTT Technical Research Centre of Finland, sixth edition.

[16] Chin Ding Ang. Investigation of a computationally efficient multi-scale modelling method in long tunnels for FDS6. Masters thesis, Imperial College London, September 2014.

[17] Kevin McGrattan and Randall McDermott. Response to "Unexpected oscillations in fire modelling inside a long tunnel" by Ang et al. *Fire Technology*, 2021.

[18] Chin Ding Ang, Guillermo Rein, and Joaquim Peiró. Unexpected oscillations in fire modelling inside a long tunnel. *Fire Technology*, 56(5):1937–1941, 2020.

[19] C. D. Ang and J. Barnett. Numerical prediction of CO and its impact on practical application. In *Asia-Oceania Symposium on Fire Science and Technology*. IAFSS, International Association for Fire Safety Science, December 2021.

[20] C. D. Ang, J. Peiró, I. Riess, and G. Rein. Understanding and modelling the fire throttling effect in longitudinally ventilated tunnels. In *Asia-Oceania Symposium on Fire Science and Technology*. IAFSS, International Association for Fire Safety Science, December 2021.

[21] C. D. Ang, J. Peiró, I. Riess, and G. Rein. Analysis of fire throttling in longitudinally ventilated tunnels with a one-dimensional model. *Fire Technology*, 58:2925–2947, July 2022.

[22] K. McGrattan, S. Hostikka, R. McDermott, J. Floyd, and M. Vanella. *Fire Dynamics Simulator Technical Reference Guide*. NIST and VTT Technical Research Centre of Finland, sixth edition, 2021.

[23] NIST. FDS-SMV. Google Groups, 2022. https://groups.google.com/g/fds-smv.

[24] H. Ingason, Y.Z. Li, and A. Lönnermark. Runehamar tunnel fire tests. *Fire Safety Journal*, 71:134–149, 2015.

[25] U.S. Department of Transportation. *Subway Environmental Simulation Computer Program*. Federal Transit Administration, 4.1 edition, August 2001.

[26] Z. Wang, F. Jia, and E. R. Galea. Predicting toxic gas concentrations resulting from enclosure fires using local equivalence ratio concept linked to fire field models. *Fire and Materials*, 31(27-51), 2007.

[27] F. V. Lundstrom, P. V. Hees, and E. Guillaume. A review on prediction models for full-scale fire behaviour of building products. *Fire and Materials*, 41:225–244, 2017.

[28] C. L. Beyler. Major species production by diffusion flames in a two-layer compartment fire environment. *Fire Safety Journal*, 10:47–56, January 1986.

[29] International Organization for Standardization. ISO 13571 Life threatening components of fire - Guidelines for the estimation of time to compromised tenability in fires. Technical report, ISO, Switzerland, September 2012.

[30] A. Tewarson. Fully developed enclosure fires of wood cribs. *The Combustion Institute*, pages 1555–1566, 1984.

[31] I. Hansen-Bruhn and T. Richard Hull. Smoke toxicity of fire protecting timber treatments. *Fire Safety Journal*, 141, 2023.

[32] R. M. Hadden A. I. Bartlett and L. A. Bisby. A review of factors affecting the burning behaviour of wood for application to tall timber construction. *Fire Technology*, 55:1–49, 2019.

[33] A. I. Bartlett J. P. Hidalgo S. Santamaria S. Deeny F. Wiesner, L. A. Bisby and R. M. Hadden. Structural capacity in fire of laminated timber elements in compartments with exposed timber surfaces. *Engineering Structures*, 179:284–295, 2019.

[34] T. Gernay. Fire resistance and burnout resistance of timber columns. *Fire Safety Journal*, 122, 2021.

[35] N. Johansson and M. Ekholm. Variation in results due to user effects in a simulation with FDS. *Fire Technology*, 54:97–116, 2018.

[36] A. Rafinazari and G. Hadjisophocleous. A study of the effect of make up air velocity on the smoke layer height with symmetric openings in atrium fires. *Fire Technology*, 54:229–253, 2018.

[37] Henry Petroski. *Success through failure: The paradox of design.* Princeton University Press, 2018.

[38] A. Lönnermark, B. Persson, and H. Ingason. Pulsations during large-scale fire tests in the Runehamar tunnel. *Fire Safety Journal*, 41(5), 2006.

[39] C. D. Ang, G. Rein, J. Peiró, R. Harrison, and I. Vermesi. Investigation of a computationally efficient multi-scale modelling method in longitudinally ventilated tunnels for FDS6.1. In *Fire and Evacuation Modelling Technical Conference*. Thunderhead Engineering, 2016.

[40] P.M. Gresho and R.L. Lee. Don't suppress the wiggles – they're telling you something! *Computers and Fluids*, 9(2):223–253, June 1981.

[41] I.M. Vermesi, G. Rein, F. Collella, 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*, 64, 2017.

[42] A. Vaitkevicius, F. Colella, and R. Carvel. Investigating the throttling effect in tunnel fires. *Fire Technology*, 52:1619–1628, 2016.

[43] R. McDermott. Discussion about choice of FDS parameters. Personal Communication, 2018.

[44] Architectural Institute of Japan. AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, 96:1749–1761, 2008.

[45] OECD. *Best practice guideliens for the use of CFD in nuclear reactor safety application*. Organisation for Economic Co-operation and Development Nuclear Energy Agency, 2015.

[46] H. Ingason, Y.Z. Li, and A. Lönnermark. *Tunnel fire dynamics*. Springer, 2015.

[47] R.E. Greuer. Influence of mine fires on the ventilation of underground mines. Open file report 74-73, US Bureau of Mines, 1973. USBM Contract Report No SO122095.

[48] F. Collela, G. Rein, V. Verda, and R. Borchiellini. Multiscale modelling of transient flows from fire and ventilation in long tunnels. *Computers and Fluids*, 51:16–29, July 2011.

[49] C.C. Hwang and R.F. Chaiken. Effect of duct fire on the ventilation velocity. Report of Investigations 8311, US Bureau of Mines, 1978.

[50] C.K. Lee, R.F. Chaiken, and J.M. Singer. Interaction between duct fires and ventilation flow: an experimental study. *Combustion Science and Technolog*, 20(1–2):59–72, July 1979.

[51] C.D. Litton, M.I. DeRosa, and J.-S. Li. Calculating fire throttling of mine ventilation airflow. Report of investigations 9076, US Bureau of Mines, 1987.

[52] Rickard Hansen. Mass flow during fire experiments in a model-scale mine drift with longitudinal ventilation. *Mining Technology*, 129(2):68–81, 2020.

[53] J. Bickel, T. Kuesel, and E. King, editors. *Tunnel Engineering Handbook*. Kluwer Academic Publishers, second edition, 1996.

[54] I.E. Idelchik. *Handbook of hydraulic resistance*. Jaico Publishing House, third edition, 2007.

[55] S. Genic, I. Arandjelovic, P. Kolendic, P. Jaric, M. Budimir, and V. Genic. A review of explicit approximations of Colebrook's equation. *FME Transactions*, 39:67–71, 2011.

[56] C. Fleming, G. Clark, K. Meeks, and T. Wicht. The treatment of the throttling effect in incompressible 1D flow solvers. In *8th International Conference on "Tunnel Safety and Ventilation"*, Graz, June 12–14 2016.

[57] Futoshi Tanaka, Kohei Takezawa, Yuji Hashimoto, and Khalid A.M. Moinuddin. Critical velocity and backlayering distance in tunnel fires with longitudinal ventilation taking thermal properties of wall materials into consideration. *Tunnelling and Underground Space Technology*, 75:36–42, 2018.

[58] Y.Z. Li and H. Ingason. The maximum ceiling gas temperature in a large tunnel fire. *Fire safety journal*, 48:38–48, 2012.

[59] Österreichische Forschungsgesellschaft Straße - Schiene - Verkehr. Tunnel ventilation systems – basic principles. Technical Report RSV 09.02.31, Österreichische Forschungsgesellschaft Straße-Schiene-Verkehr, 2014.

[60] B. J. McCaffrey. Purely buoyant diffusion flames: Some experimental results. Technical Report MBSIR 79-1910, National Bureau of Standards, Center for Fire Research, October 1979.

[61] Dougal Drysdale. *An introduction to fire dynamics*. Wiley, 3rd edition, 2011.

[62] C. D. Ang and J. Peiró. TE1D one-dimensional throttling model framework. Zenodo web page, 2022. https://doi.org/10.5281/zenodo.5831566.

[63] H. Ingason and Y.Z. Li. Model scale tunnel fire tests with longitudinal ventilation. *Fire Safety Journal*, 45(6–8):371–384, 2010.

[64] Liming Li, Sen Li, Xuegui Wang, and Heping Zhang. Fire-induced flow temperature along tunnels with longitudinal ventilation. *Tunnelling and Underground Space Technology*, 32:44–52, 2012.

[65] Y.Z. Li. Study of fire and explosion hazards of alternative fuel vehicles in tunnels. *Fire Safety Journal*, 110:102871, 2019.

[66] Peng Lin, Yuan-Yuan Xiong, Cong Zuo, and Jia-Kang Shi. Verification of similarity of scaling laws in tunnel fires with natural ventilation. *Fire Technology*, 57(4):1611–1635, 2021.

[67] Liang Gong, Lin Jiang, Shuoyu Li, Na Shen, Yuchun Zhang, and Jinhua Sun. Theoretical and experimental study on longitudinal smoke temperature distribution in tunnel fires. *International Journal of Thermal Sciences*, 102:319–328, 2016.

[68] L. Anderson. The biggest tunnelling projects around the world in 2021, November 2021. https://www.international-construction.com/news/the-biggest-tunnelling-projects-around-the-world-in-2021/8013496.article.

[69] G. E. P. Box. Science and statistics. *Journal of the American Statistical Association*, 71(356):791–799, May 1976.

[70] F. Colella, G. Rein, R. Borchiellini, R. Carvel, J.L. Torero, and V. Verda. Calculation and design of tunnel ventilation systems using a two-scale modelling approach. *Building and Environment*, 44:2357–2367, 2009.

[71] Novenco. Tunnel jet fans. Webpage, 2022. https://www.novenco-building.com/media/1336/novax-tunnel-cat-gb-mu16159-0322.pdf.

[72] F. Colella, G. Rein, R. Borchiellini, and J.L. Torero. A novel multiscale methodology for simulating tunnel ventilation flows during fires. *Fire technology*, 47(1):221–253, January 2011.

[73] Victoria and Albert Museum. Victoria and Albert Museum, September 2022. https://www.vam.ac.uk/articles/computers-and-the-sydney-opera-house.