

I-70 East Project, Denver, CO. Partial Covered Lowered Alternative CFD Modeling Study of Emergency Ventilation

Atkins North America

ATKINS

16 September 2015

Plan Design Enable

Table of contents

Chapter	Pages
1. Introduction	6
2. Scope	6
2.1. Purpose	6
2.2. Objective	6
2.3. Limitations	6
3. CFD Modeling	6
3.1. Methodology	6
3.2. Model Geometry	7
3.3. Model Assumptions and Boundary Conditions	9
4. Results	15
4.1. IDA Fire Case	15
4.2. IDA Incoming Air Temperature Effects	16
4.3. CFD Results	16
5. Conclusions	23
5.1. Comparisons	23
5.2. Ventilation CRD assessment	23
6. References	24

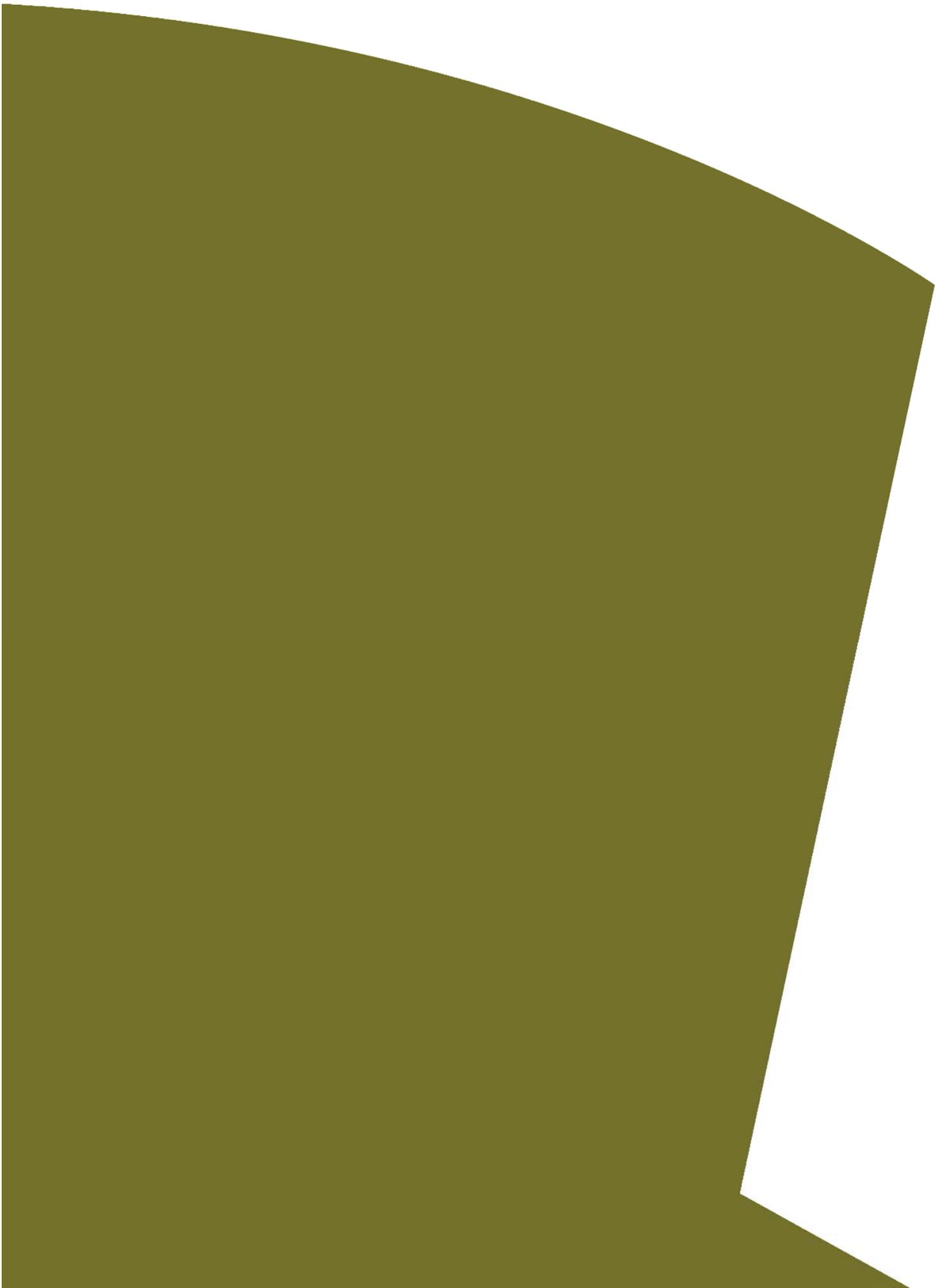
Tables

Table 3-1 Determination of three 'model vehicle types' from the prescribed 'source type hour fraction'.	12
Table 4-1 IDA predicted flow rates and condition resulting from a mid-cover 30 MW fire.	16
Table 5-1 Steady portal flow rates and temperatures, determined by IDA (1D) and CFD (3D).	23

Figures

Figure 3-1	CFD model of I-70 cover westbound bore, shown in plan, inclusive of jet fans, backed-up traffic, and the incident vehicle approximately mid-way along the cover.	8
Figure 3-2	CFD model of I-70 cover westbound bore, east end elevation looking toward the exit portal. Operating jet fans (14) shaded white. Redundancy provided by (2) jet fans shaded black.	8
Figure 3-3	CFD model of I-70 cover westbound bore, viewed in perspective from below, looking west. Converging wall shown adjacent to westbound on-ramp (indicated by steps on the right of the image).	8
Figure 3-4	CFD model representation of jet fan niche, in westbound bore.	9
Figure 3-5	History of fire Heat Release Rate (HRR) from incident vehicle.	10
Figure 3-6	Perspective view of model vehicle train, with gap dimensions highlighted.	12
Figure 3-7	Model vehicle 'train' used in CFD model.	13
Figure 3-8	Repeated model vehicle trains, staggered between lanes.	13
Figure 3-9	Distribution of lanes, shoulders, and gaps across the highway, looking west from the entry portal.	14
Figure 3-10	Two views looking west from the entry portal, in perspective, showing convergence of the on-ramp.	14
Figure 3-11	Jet fans represented as square section tubes, of equivalent internal hydraulic diameter to the 1.12 m internal diameter jet fans.	15
Figure 4-1	Tunnel ventilation performance increasing with decreasing incoming air temperature	16
Figure 4-2	Fire generated soot, viewed in plan. Ventilation full capacity, Fire HRR 30 MW.	17
Figure 4-3	Fire generated soot, looking west from within vehicle 3 rows back from fire. Ventilation full capacity, Fire HRR 30 MW.	17
Figure 4-4	Tracer particles, viewed from the exit portal. Ventilation full capacity, Fire HRR 30 MW.	17
Figure 4-5	Histories of total smoke mass, nitrogen mass, and oxygen mass under the cover.	18

Figure 4-6	History of water vapour mass under the cover.	19
Figure 4-7	History of carbon dioxide mass under the cover.	19
Figure 4-8	Histories of soot mass, carbon monoxide mass, and fuel mass under the cover.	20
Figure 4-9	60°C isotherm in purple, sliced to show temperature contours above 60°C. Ventilation full capacity, Fire HRR 30 MW.	21
Figure 4-10	Isotherms viewed in plan. Ventilation full capacity, Fire HRR 30 MW.	21
Figure 4-11	Smoke temperature contours at various cross-sections, on restricted temperature scale.	22



Executive summary

A CFD modeling study using the industry standard Fire Dynamics Simulator (FDS), has been undertaken to simulate smoke migration from a specific fire scenario under the I-70 cover. The ventilation Concept Reference Design (CRD) based on jet fans located just in from the entry portal, has been determined by prediction using 1D Modeling Software IDA to be sufficient to prevent back-layering of smoke and hot gases, in order to protect users of the covered highway, and emergency services in attendance.

The westbound bore with a negative grade in the direction of travel, offers the greatest challenge to ventilate should a fire emergency materialise. A particularly challenging fire scenario using a convective Heat Release Rate (HRR) of 30 MW based on the design fire, and situated close to the midpoint of the cover is implemented. The fire scenario also includes stationary traffic backed-up behind the incident vehicle, and an adverse wind pressure exerted at the exit portal. Both are designed to offer extra impedance against the ventilation induced by a row of jet fans, mounted in a ceiling niche.

The study includes a brief justification for the selection of the particular CFD program, and description of the assumptions and boundary conditions used. Initial development of the CFD model is facilitated by an additional IDA calculation excluding the fire, so that the jet fan representation in the CFD model could be gauged to impart the correct impulse to the air under the cover. IDA calculation results demonstrate that the tunnel ventilation system performance increases with decreasing incoming air temperature and validates the selection of incoming air temperature at 38°C in the CFD simulations.

Results from the CFD simulation of the fire scenario provides evidence by demonstration, that the ventilation CRD described in [1], is capable of preventing back-layering of smoke. Comparison of the predicted ventilation flow rates from the IDA simulation, and from the CFD simulation, presents some quantitative differences.

1. Introduction

Atkins North America (ANA) has been commissioned by the Colorado Department of Transportation (CDOT) to develop the scheme for the I-70 East Project, an upgrade to the interstate highway I-70 in Denver, Colorado, called the Partial Cover Lowered Alternative (PCLA). The upgrade involves lowering the highway between Brighton Blvd and Colorado Blvd, and building a cover over part of the lowered highway, starting marginally east of Clayton Street and ending marginally west of Columbine Street.

As part of the scheme, Atkins Ltd (AUK) has been commissioned by ANA to conduct a Computational Fluid Dynamics (CFD) modeling study of smoke control provisions under the short cover, in order to confirm performance of the ventilation Concept Reference Design (CRD) for a fire scenario based on the prescribed 'Design Fire'.

Prior to this study, AUK developed the ventilation CRD using performance analysis based on application of a 1D tunnel ventilation computer program called the IDA Road Tunnel Ventilation (IDA) simulator. The CFD study delivers detailed results describing the condition under the cover.

2. Scope

2.1. Purpose

The purpose of this CFD modeling study is to demonstrate that the ventilation CRD described in the ventilation and fire life safety report [1] will be able to prevent back-layering of smoke from a prescribed fire emergency scenario contained in the I-70 cover, based on 30 MW convective portion of the design fire Heat Release Rate (HRR), as determined in the first instance by 1D analysis. Hence, this is a verification of the Critical Velocity (CV) calculation used in the 1D analysis.

2.2. Objective

The objective of this CFD modeling study is to compare the mid-cover fire simulation results with results from the same configuration using the IDA simulator, and verify that the back-layering of smoke and hot gases can be controlled.

2.3. Limitations

Findings in this report are based on CFD model simulation results. CFD models are founded on mathematical description of the physical mechanisms in the field of fluid dynamics, heat transfer and chemical reactions. As with all mathematical descriptions of physical processes, it is accepted that there will be an error tolerance or inaccuracy associated with that description, and with any additional modeling assumptions made.

3. CFD Modeling

3.1. Methodology

Prior to this study, ventilation of the I-70 cover has been simulated using a 1D approach with IDA. Analysis of the IDA steady state results has provided an assessment that the ventilation CRD will control smoke from a fire scenario based on the convective portion of the design fire, so that passengers in the vehicles upstream of the fire incident will not be exposed to excessive heat or toxic gases.

The analysis of IDA results was conducted using the critical velocity (CV) calculation that quantifies the threshold rate of ventilation air, necessary to prevent back-layering of the resulting smoke over the vehicles backed-up behind the fire incident. A limitation of 1D modeling is that flow features that are strongly 3D, such as back-layering, are not represented. A means of verifying the 1D analysis is used here by 3D modeling,

often referred to as field modeling. Additionally, the field model used in this study also accounts for transient flow behaviour.

3.1.1. Field Modeling using CFD

Computational field modeling of a fluid dynamic system is substantially more complex than 1D modeling because it accounts fundamentally for 3D spatial mechanisms, and effects, that are inherently interdependent. The category of fluid dynamics modelled here is incompressible, buoyancy driven, turbulent flow of smoke laden air (from now on referred to as smoke), with gaseous reactions. To solve the complex, non-linear equations that describe 3D transport of smoke and heat, a computationally intensive numerical approach is required. The category of field models used for this approach to simulation of fluid dynamic systems is called CFD. The specific CFD program used in this study is the Fire Dynamics Simulator (FDS) described in Section 3.1.2.

3.1.2. Fire Dynamics Simulator – FDS

The Fire Dynamics Simulator (FDS) is a CFD program produced by the National Institute of Standards and Technology (NIST) of the US Department of Commerce, in co-operation with the Technical Research Centre (VTT) of Finland, specifically for computer simulation of fire driven smoke flow. FDS solves numerically a Large Eddy Simulation (LES) form of the Navier-Stokes equations appropriate for low speed, thermally driven flow, with an emphasis on smoke and heat transport from fires. For source documentation on FDS, see the user guide [2], the mathematical basis guide [3], the verification guide [4], and the validation guide [5].

FDS has been widely adopted by the fire safety/protection engineering community as the industry standard tool for comprehensive and detailed analysis of combustion from fire, and the subsequent transport of smoke and heat inside buildings, and for the outside environment. Its uses include:

- Investigation of real fire incidents;
- Demonstrating performance of fire safety measures in design of new-build structures;
- Consequence analyses for emergency evacuation of people.

A companion program called Smokeview is used to display/visualise output from FDS. For source documentation on Smokeview, see the user guide [6], the technical reference guide [7], and the verification guide [8].

3.2. Model Geometry

3.2.1. Covered Highway

The I-70 East Project PCLA consists of a less than 1,000 ft cover constructed over the lowered I-70. Drawings used in the ventilation and fire life safety analysis reported in [1], are also used as the basis of this modeling study to ensure compatibility in results. They include preliminary drawings for: a typical cross-section through the cover [9]; the road profile [10]; and plan views [11]. Dimensions taken from these drawings were used in the first instance to inform the IDA analysis, and for consistency, have been used to inform this CFD modeling study. For clarity, the actual dimensions used in the CFD model are annotated in Figure 3-1, Figure 3-2, and Figure 3-3 below.

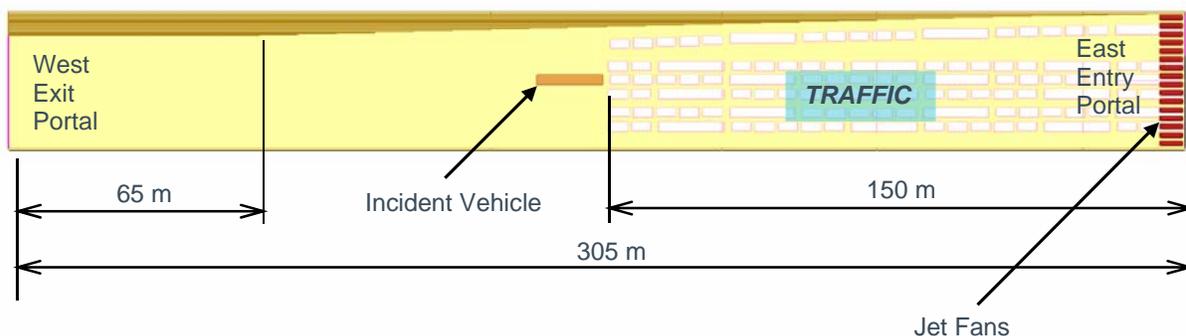


Figure 3-1 CFD model of I-70 cover westbound bore, shown in plan, inclusive of jet fans, backed-up traffic, and the incident vehicle approximately mid-way along the cover.

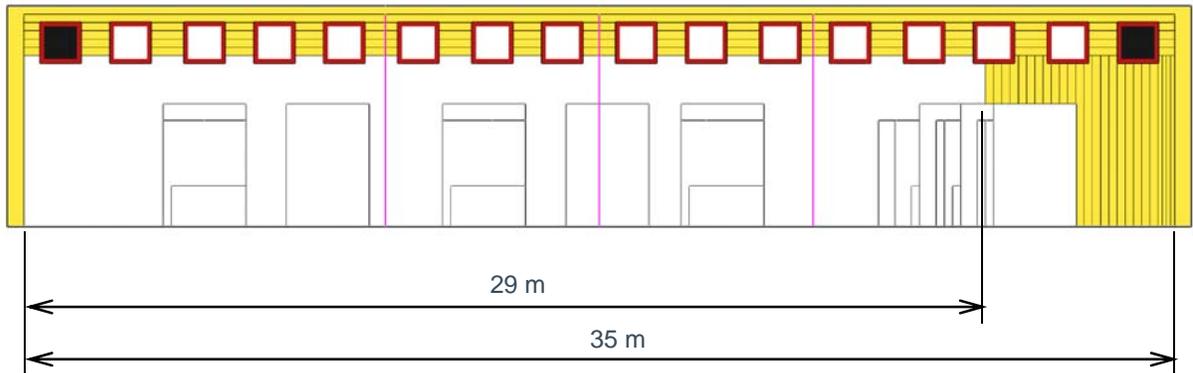


Figure 3-2 CFD model of I-70 cover westbound bore, east end elevation looking toward the exit portal. Operating jet fans (14) shaded white. Redundancy provided by (2) jet fans shaded black.

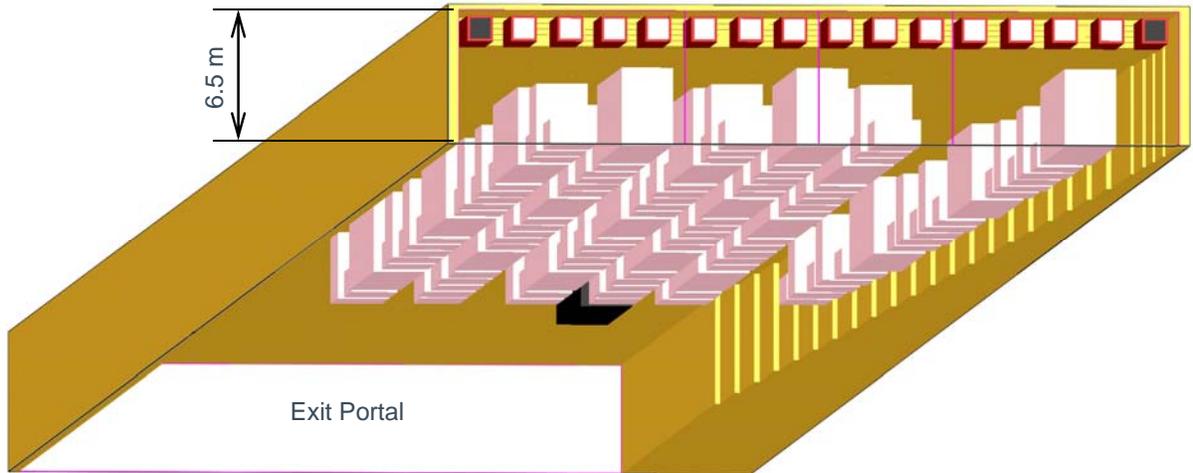


Figure 3-3 CFD model of I-70 cover westbound bore, viewed in perspective from below, looking west. Converging wall shown adjacent to westbound on-ramp (indicated by steps on the right of the image).

3.2.2. Jet Fan Niche

The jet fan niche has been sized to accommodate a single row of 16 jet fans as defined in the ventilation CRD in [1], accounting for the low height of the cover ceiling. The niche dimensions are prescribed in imperial units. The niche depth is 4 ft, rising from the ceiling height of 17.5 ft above the road surface. The niche length is 24 ft starting at the entry portal, followed by a further 9 ft of uniform slope between the niche and the ceiling. Figure 3-4 shows the niche, and a jet fan as represented in the CFD model, annotated with dimensions in SI units that are adjusted a small amount for ease of modeling.

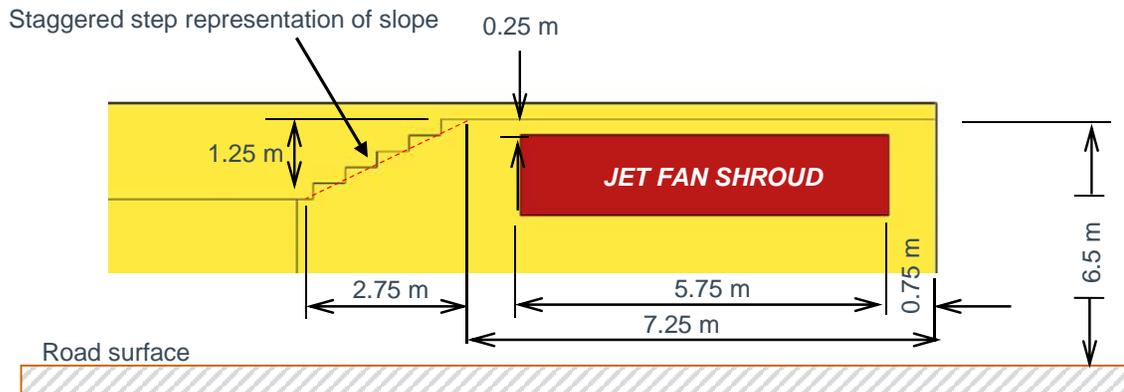


Figure 3-4 CFD model representation of jet fan niche, in westbound bore.

3.2.3. Westbound On-Ramp

According to the road profile in [3], the westbound on-ramp shares the highway grade for the entire shared length under the cover. Hence there is no difference in elevation for the respective lanes.

3.3. Model Assumptions and Boundary Conditions

3.3.1. Altitude

As Denver is situated at a high enough altitude to necessitate use of a non-default atmospheric pressure, this is accounted for by using a reduced atmospheric pressure of 84.3 kPa for the entire model domain. The ambient air temperature is 38°C and, ambient air density 0.945 kg/m³.

3.3.2. Smoke and Initial Condition

For this study, 'smoke' refers to the gaseous mixture of:

- Fuel vapour
- Atmospheric nitrogen, N₂
- Atmospheric oxygen, O₂
- Carbon dioxide, CO₂
- Carbon monoxide, CO
- Water vapour, H₂O
- Soot (assumed to be pure carbon and continuously suspended in the smoke mixture)

Hence, by way of example, dry air which is: nitrogen 75.47%; oxygen 23.20%; carbon dioxide 0.046%, and (hydrogen, argon, neon, helium, krypton, xenon) 1.28% by mass at STP, would be represented as 'smoke' with mass fractions: 0.7645 kg(N₂)/kg(smoke); 0.2350 kg(O₂)/kg(smoke), and 0.0005 kg(CO₂)/kg(smoke), normalised to discount the trace molecules that are not solved for. For the purpose of this study, dry air will from now on be described as clean smoke. Pure fuel vapour would be described as 'smoke' with mass fraction 1 kg(fuel)/kg(smoke).

A CFD analysis requires the prescription of an initial field of values for all primary variables solved. Primary variables are the smoke scalar and vector quantities solved by FDS using the transient, 3D, conservative transport equations. The primary variables that describe the flow and condition of 'smoke' in this study are:

- Smoke pressure (fluctuation relative to atmospheric)
- Three Cartesian components of smoke velocity -
 - Longitudinal
 - Lateral
 - Vertical
- Smoke temperature

- Mass fraction of the gaseous constituents listed above
- Mass fraction of suspended soot, also listed above

3.3.3. Combustion and the Fire Scenario

The fire (combustion source) is specified as a burning vehicle, in this case a combination long-haul truck (defined later in

Table 3-1 as a model vehicle type 1) that has been chosen as the participating/incident vehicle. The fire is represented as a growing generation rate of a hydrocarbon fuel (the fuel vapour) uniformly spread over the entire outside surface (five exposed faces) of the incident vehicle. The purpose as stated in Section 2 is to demonstrate that the ventilation CRD will be able to prevent back-layering of smoke and hot gases from a prescribed fire emergency scenario, based on the 30 MW convective portion of the design fire HRR. The fire growth rate is not of significance to the outcome of this study. It is necessary however to implement an exceptionally high fire growth rate to develop the simulation quickly but without imposing an instantaneous large HRR that could compromise the numerical stability of the calculation. For this reason, the fire is assigned a t-squared growth rate that enables it to develop from 0 to 30 MW in a period of 60 s as illustrated in Figure 3-5, after which it remains constant at 30 MW. This is an unrealistically high rate but is used here for modeling convenience.

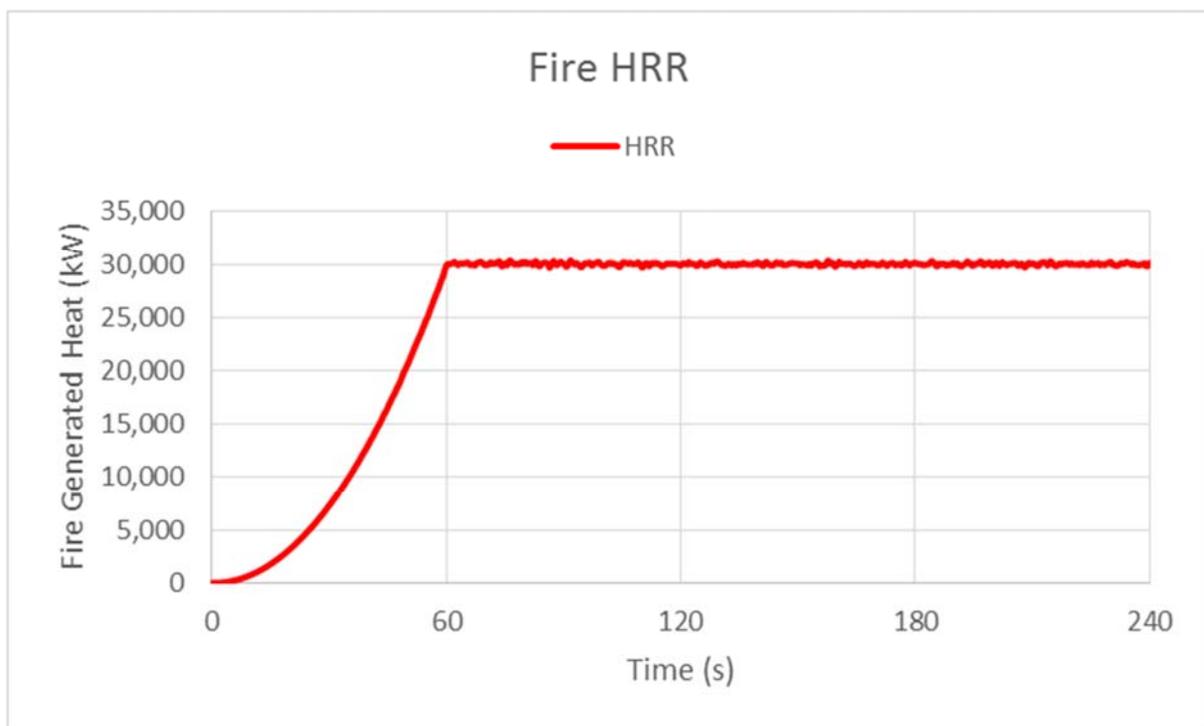


Figure 3-5 History of fire Heat Release Rate (HRR) from incident vehicle.

3.3.4. Portals and Wind

The entry and exit portals shown in Figure 3-1, constitute the limits of the computational domain. Pressure boundaries representative of the external/ambient condition are defined separately at each portal. This allows for the possibility of counter flow of smoke at portals when the pressure difference across a portal changes sign along its section.

For this study, an adverse wind pressure is also accounted for by imposing a zero external pressure at the entry portal and a positive external pressure at the exit portal. An exit portal pressure of 20 Pa is used as approximately representative of the dynamic pressure exerted by a 6 m/s wind acting normal to, and directed at the exit portal. The external ambient temperature is 38°C, and all constituent scalars are held at their initial values.

3.3.5. Grade and Buoyancy

The road grade in the westbound bore varies minutely between portals [3]. For this study a uniform mean grade of 1.19% is used. A consequence of this should be that buoyancy of any hot smoke will drive it to impinge on the cover ceiling, where the subsequent smoke layer will spread horizontally in all directions when not ventilated, with a preference to travel longitudinally uphill toward the entry portal. This is taken into account by the model.

3.3.6. Cover Ceiling

The cover ceiling is assumed to be flat and equidistant from the road surface for the entire length of the covered highway, with the exception of the jet fan niche. Hence the cover ceiling follows the average grade of the road. The ceiling material and fixtures are assumed non-combustible, fire resistant, and it is assumed the ceiling will remain intact for all conditions. As illustrated in Figure 3-4, the niche follows the same grade as the cover ceiling. The niche is assumed to have the same material specification as the cover ceiling.

3.3.7. Traffic and Lanes

For the purpose of this study, the traffic is stationary due to the fire incident precipitating closure of the highway under the lid, and all vehicles situated downstream of the incident have exited the covered highway. This includes those vehicles both sides of the incident vehicle, illustrated by Figure 3-1. All of the remaining vehicles backed-up behind the incident vehicle are stationary, and treated in the CFD model as inert, generating no heat, and no fumes. It is assumed that drivers and passengers have either been instructed to leave their vehicles and exit the cover via the entry portal, or that they have simply been instructed to switch off their engines. The vehicles are however assumed to remain at the initial cover temperature of 38°C.

The physical presence of the vehicles is represented simplistically by multiple individual rectangular blocks, constrained to be inline due to marked lanes. The blocks sit flush with the road surface, a simplification that takes no account for flow under the vehicles. As the vehicles are not moving, this simplification is not expected to be a significant influence on smoke flow.

The representation of traffic vehicle mix in the CFD model conforms approximately to the 'source type hour fraction' prescribed in a MOVES Link Source Type file, supplied by ANA [12], and replicated in Table 3-1 below. The thirteen vehicle names considered for I-70 were sifted in Table 3-1 using the following process, into three groups of similar height, width, and length:

- Motorcycles not included
- All vehicles of length 18 m, are also 2.5 m wide, and have a narrow height range
- Normalised - source type hour fraction for model vehicle type 1 is 50.49% (approximately 3 in 6)
- Normalised - source type hour fraction for model vehicle type 2 is 33.07% (approximately 2 in 6)
- Normalised - source type hour fraction for combined model vehicle types 3,4,5 is 16.44% (approximately 1 in 6)

The final column in Table 3-1 shows that the grouping of source type hour fraction into three model vehicle types, conveniently enables the model vehicle types to fit a 'train' of six model vehicles, illustrated by Figure 3-6.

Table 3-1 Determination of three ‘model vehicle types’ from the prescribed ‘source type hour fraction’.

I-70 vehicle name	AASHTO vehicle name	Dimensions (m)			Source type hour fraction (%)	Model vehicle type	No. of vehicles of each type in a 6 car train
		Height	Width	Length			
Motorcycle	NA				3.45		
Passenger car	Passenger car	1.25	2.25	6.00	48.75	1	3
Passenger truck	Single unit truck (min height)	3.25	2.50	6.00	31.93	2	2
Light commercial truck	Single unit truck (max height)	4.00	2.50	18.00	10.29	3	1
Intercity bus	Intercity bus (motor coaches)	3.75	2.50	18.00	0.39	4	
Transit bus	City transit bus	3.25	2.50	18.00	0.26	5	
School bus	Conventional school bus	3.25	2.50	18.00	1.68	5	
Refuse truck	Single unit truck (max height)	4.00	2.50	18.00	0.04	3	
Single unit short-haul truck	Single unit truck (max height)	4.00	2.50	18.00	1.60	3	
Single unit long-haul truck	Single unit truck (max height)	4.00	2.50	18.00	0.10	3	
Motorhome	Motorhome	3.75	2.50	18.00	0.16	4	
Combination short-haul truck	Interstate semi-trailer	4.00	2.50	18.00	0.77	3	
Combination long-haul truck	Interstate semi-trailer	4.00	2.50	18.00	0.59	3	

Some judgement is used in selection of the respective gaps between each vehicle in the vehicle ‘train’ shown in Figure 3-6, generally being 1 m, except for ahead of and behind the largest model vehicle type, where the gaps are 2 m. This also conveniently fits a model vehicle ‘train’ length of 50 m, that is implemented repeatedly in the CFD model, for each lane as illustrated by Figure 3-8.

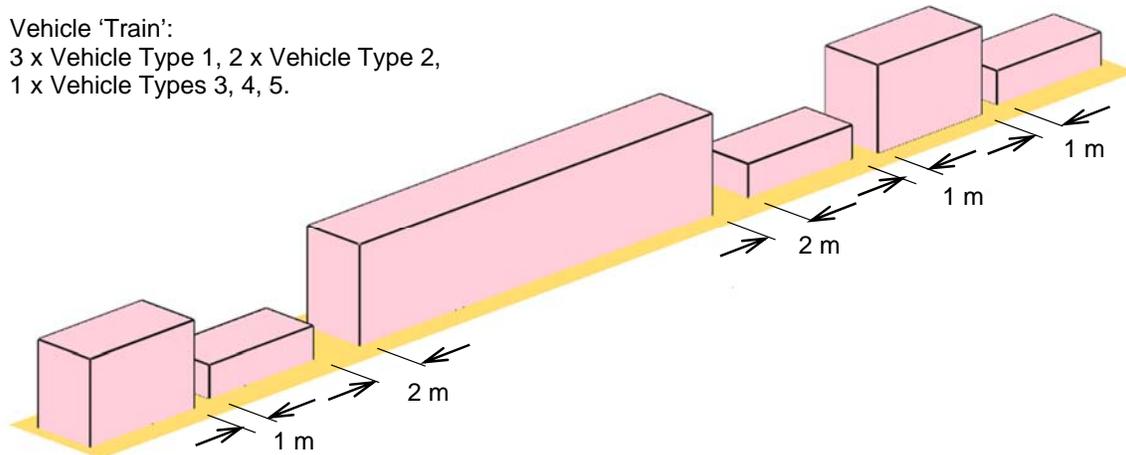
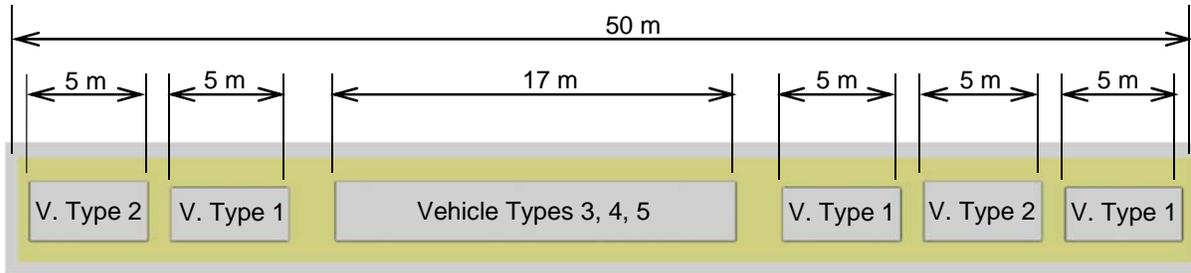


Figure 3-6 Perspective view of model vehicle train, with gap dimensions highlighted.

Dimensions of the model vehicle ‘train’, and the total length of the vehicle ‘train’ plus gaps, are annotated in Figure 3-7 below. The sequence from front to back of vehicle types selected arbitrarily for the model vehicle ‘train’ is: type 2; type 1; types 3,4,5; type 1; type 2; type 1, also annotated in Figure 3-7.



Vehicle Type 1 (2.25 m wide, 1.25 m high), Vehicle Type 2 (2.50 m wide, 3.25 m high),
Vehicle Types 3, 4, 5 (2.50 m wide, 3.75 m high)

Figure 3-7 Model vehicle 'train' used in CFD model.

The lines of model vehicle 'trains' are then staggered between neighbouring lanes, as illustrated by Figure 3-8. This Figure also shows how densely packed the stationary traffic backed-up behind the incident vehicle is, an impression not easily visualised through numbers alone.

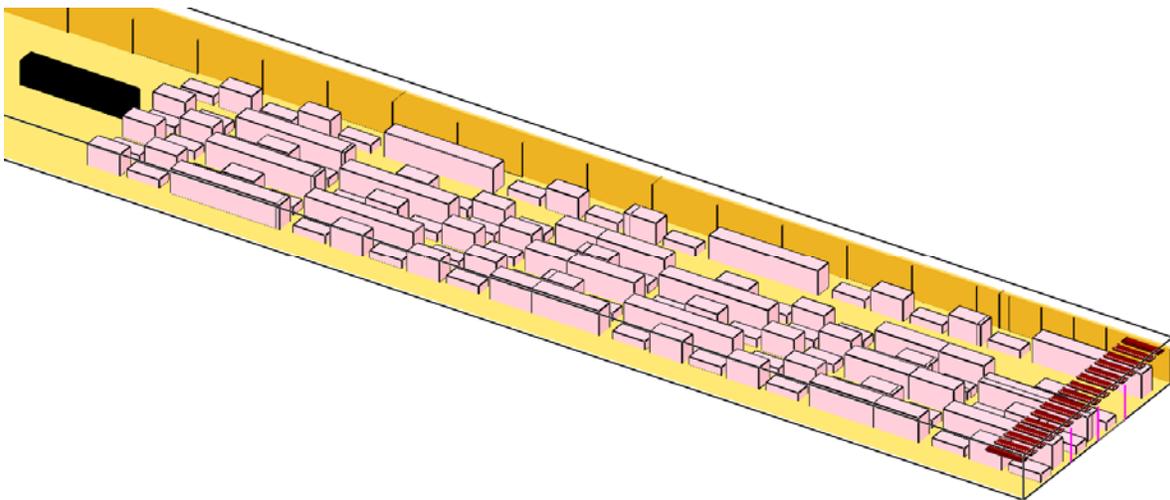


Figure 3-8 Repeated model vehicle trains, staggered between lanes.

For the 'PCL-ML' Full Build option [12] represented by the CFD model, there are a total of six lanes, as illustrated by Figure 3-9. When viewing west from the entry portal, the lanes from left to right are:

- A 12 ft shoulder between the cover dividing wall and the first Managed Lane (ML)
- Two Managed Lanes
- A 4 ft buffer gap between the second Managed Lane and the first General Purpose Lane (GP)
- Three General Purpose Lanes
- A variable gap, diminishing between the third GP and the westbound 'on-ramp'
- The westbound 'on-ramp'
- An 8 ft shoulder between the 'on-ramp' and the cover side wall

The variable gap that separates the on-ramp from the lane indicated by GP3, reduces from a maximum of 5.78 m to nothing by approximately three-quarters of the length of the cover in the direction of travel. This means that the total highway width commencing at the entry portal is approximately 35 m, reducing to approximately 29 m at the three-quarter point, and then remaining at 29 m for the remaining quarter of the cover length.

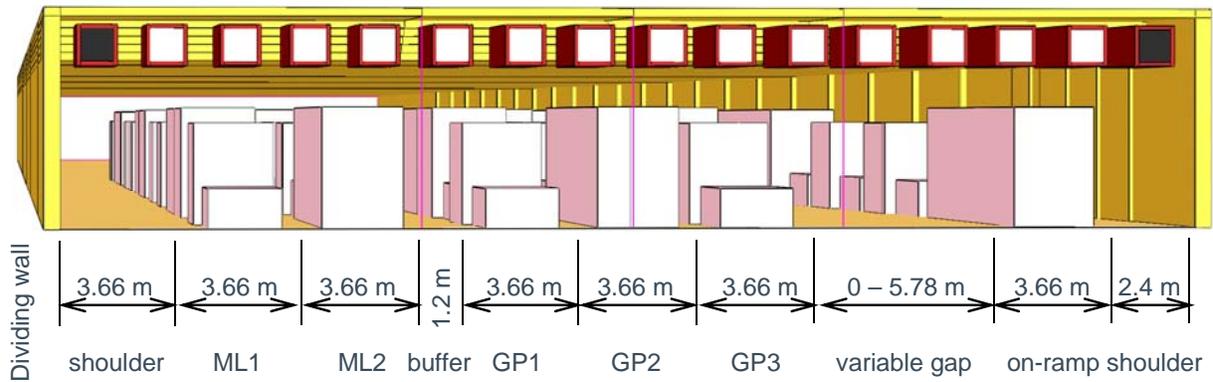


Figure 3-9 Distribution of lanes, shoulders, and gaps across the highway, looking west from the entry portal.

The convergence of the on-ramp to become the sixth lane is shown by two images in perspective, again viewing from the entry portal looking west, in Figure 3-10 below.

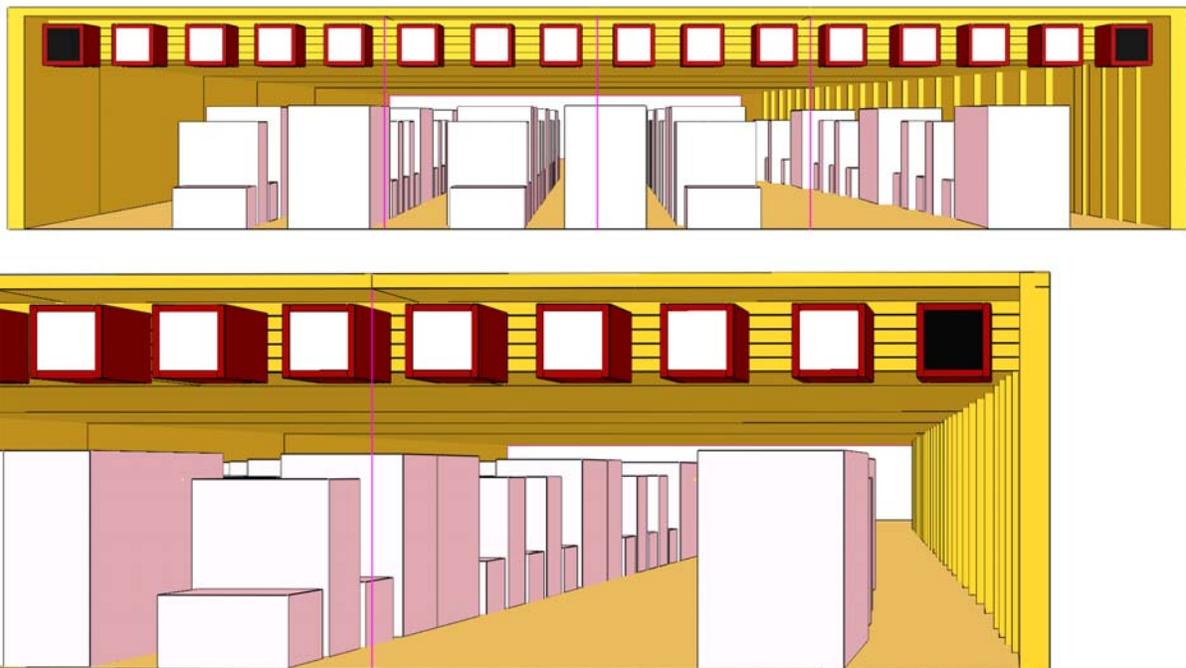


Figure 3-10 Two views looking west from the entry portal, in perspective, showing convergence of the on-ramp.

3.3.8. Jet Fan Representation

Jet fans are straightforward devices, designed to impart longitudinal momentum to the surrounding air. When used with sufficient capacity inside the cover, the resulting impulse will induce a longitudinal flow of outside air that sweeps the entire cover volume from entry portal to exit portal. The purpose for doing this is:

- To clear vehicular pollution that would otherwise accumulate and become hazardous during times when the traffic is stopped or slow moving;
- During a fire incident under the cover, to force the combustion products inclusive of soot particles, and the heat, away from the occupied vehicles backed-up behind the fire.

During normal operations, sufficient ventilation is usually induced by the piston-effect generated by the moving traffic, and the jet fans are switched off.

Jet fans are represented in the model in a simplified way, made necessary by modeling constraints with FDS, as shown in Figure 3-11.

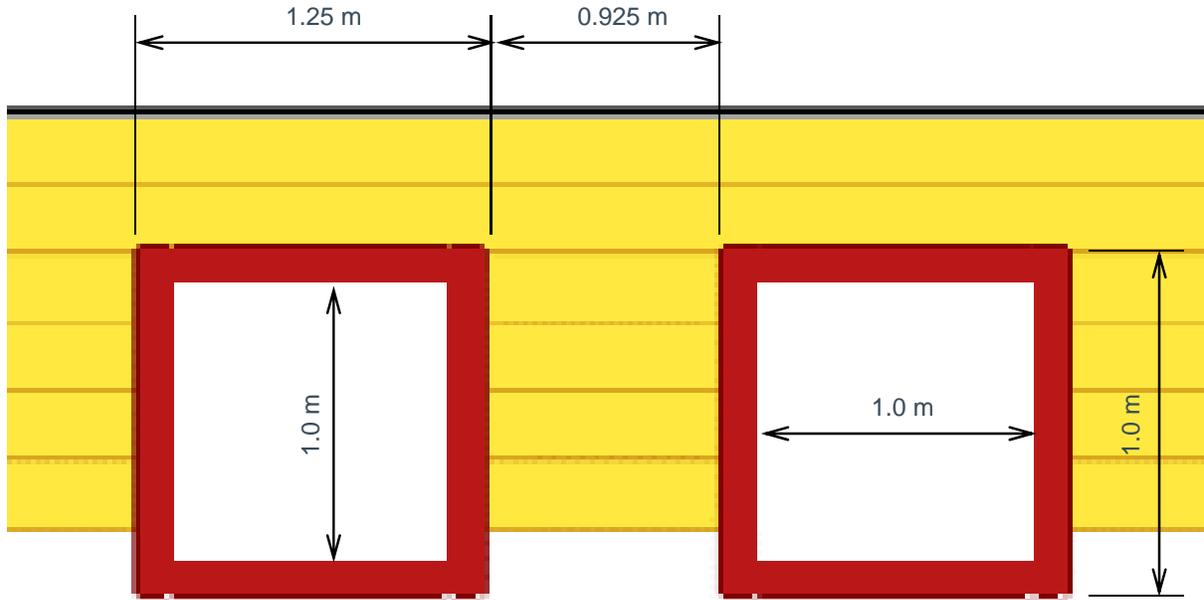


Figure 3-11 Jet fans represented as square section tubes, of equivalent internal hydraulic diameter to the 1.12 m internal diameter jet fans.

As the niche in which the jet fans are located is necessarily short, the separation between the jet fan exit and the end of the niche is also quite short, 0.75 m at the closest point, increasing to 3.5 m at the ceiling height. The real niche end is sloped so as to reduce momentum losses caused by jet impingement. In the model, the sloping niche end can only be represented by a sequence of staggered step changes in the profile of the ceiling, illustrated by Figure 3-4.

To account for the stepped ceiling, a 1D reference case is produced using the IDA simulator but without a fire. The purpose for the reference case is to be a comparator for the induced ventilation flow predicted by the CFD model, also without a fire. The jet fan velocities in the CFD model are adjusted until the induced ventilation flow is sufficiently close to that predicted in the reference case, to conclude that they represent the same gross effect under the cover. In both cases, the traffic, and adverse portal pressure are still present.

The 1D reference case predicted an induced ventilation flow rate of 557 m³/s through the cover. Gradual adjustment of the CFD model jet fan velocities resulted in an induced ventilation flow rate of 541 m³/s through the cover, an acceptable under-prediction of 3%. This jet fan representation is adopted for ventilating the cover during the fire scenario.

4. Results

4.1. IDA Fire Case

The IDA model of the I-70 cover westbound bore, is used to simulate CRD ventilation of a mid-cover 30 MW fire, with a 20 Pa adverse pressure at the exit portal. The resulting portal flows and temperatures are summarised in Table 4-1 below.

Table 4-1 IDA predicted flow rates and condition resulting from a mid-cover 30 MW fire.

Entry (east) Portal	Exit (west) Portal
Ambient (38°C) air enters the cover at 361 m ³ /s	Smoke at an elevated temperature (119°C) leaves the cover at 455 m ³ /s

4.2. IDA Incoming Air Temperature Effects

The 3D CFD simulation was performed with an incoming ambient air temperature of 38°C as this represents the most challenging scenario for the longitudinal tunnel ventilation system concept when the balance of jet fan thrust and opposing buoyancy forces and frictional losses is considered. This is demonstrated in the results of analysis presented below in Figure 4-1.

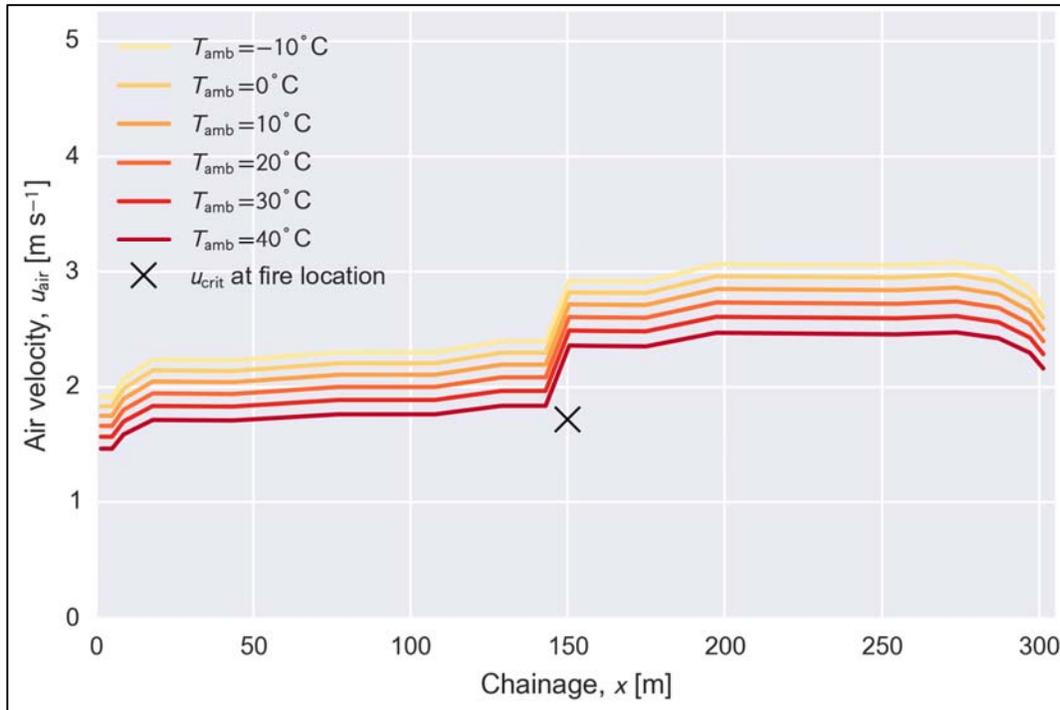


Figure 4-1 Tunnel ventilation performance increasing with decreasing incoming air temperature

We note that with decreasing ambient temperature the predicted performance of the tunnel ventilation increases as shown by the increasing air velocity to prevent the back layering of smoke and hot gases from a fire, of the same magnitude, within the covered lid section.

4.3. CFD Results

Rapid but not instantaneous introduction of the fire HRR in the first 60 s of the CFD simulation is described in Section 3.3.3, and illustrated by Figure 3-5. In parallel with introduction of the HRR, the jet fan ventilation is introduced linearly from zero to full capacity also in the first 60 s. This means that during the first 60 s, there is particular potential for smoke to back-layer given that the ventilation is not yet at full capacity. After the first 60 s, the simulation continues for a further 180 s with the fire HRR and ventilation constant at their peak values, intended to demonstrate the steady state ventilation performance.

4.3.1. Soot Distribution

From inception, a strong thermal plume forms rapidly, rising quickly to impinge on the cover ceiling. As soot is one of the products from combustion, remains entrained as part of the smoke, and is readily plotted using Smokeview, it offers an ideal visualisation medium.

At the end of the simulation, a steady flow has been established, for which a plan view of the soot distribution is shown in Figure 4-2. All soot is swept to the west, away from the stationary vehicles backed-up behind the

incident vehicle. There is also a small patch of low soot content in the smoke layer behind the incident vehicle, perhaps a sign that the wake behind the vehicle has a disruptive influence on the smoke layer. In the context of proving smoke control, this is a secondary effect.



Figure 4-2 Fire generated soot, viewed in plan. Ventilation full capacity, Fire HRR 30 MW.

The same soot distribution is shown in Figure 4-3, from the perspective of a vehicle passenger situated three vehicles upstream of the fire incident. The perspective gives little sense of longitudinal spread, although that is shown in Figure 4-2. As the soot 'front' is driven markedly downstream of the fire, the incident vehicle is partly visible to the passenger in the vehicle three rows back. This is evidence that the ventilation is strongly controlling the fate of any soot generated. There is also a sense of significant depth in the soot layer below the ceiling, enabling it to obscure the exit portal from the passengers view, even though there is also an apparent interface between dense soot and the clearer air below it.

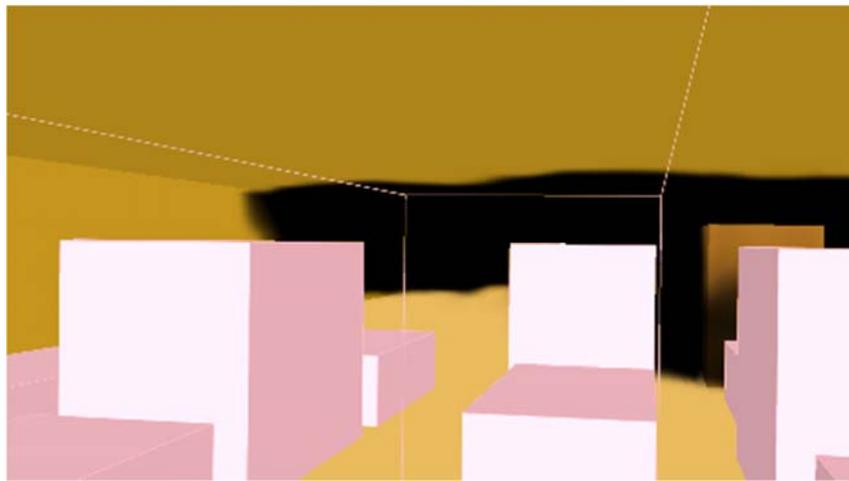


Figure 4-3 Fire generated soot, looking west from within vehicle 3 rows back from fire. Ventilation full capacity, Fire HRR 30 MW.

4.3.2. Tracer Particles

Figure 4-4 shows in purple, tracer particles viewed from the exit portal looking east toward the fire. The tracer particles are inert, weightless, and neutrally buoyant. Like the soot, they will follow the buoyancy driven smoke flow. Note that the yellow block on the left of the image is a consequence of the reduced section at the exit portal, due to the side wall following the on-ramp as it converges to form the sixth lane, as shown in Figure 3-9. This obscures the upstream tracer particles to the left hand side of each image.

This image gives an appreciation of mixing taking place between the smoke layer, and the clearer air below it, by the presence of discrete particles underneath the smoke layer. The smoke layer also appears marginally deeper adjacent to the walls.

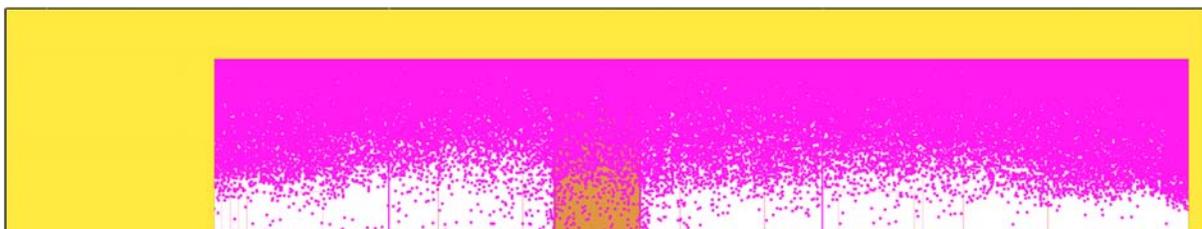


Figure 4-4 Tracer particles, viewed from the exit portal. Ventilation full capacity, Fire HRR 30 MW.

4.3.3. Smoke Constituents

The respective masses of each constituent of the air, gaseous products of combustion, and soot generated by combustion, are useful indicators of the environment under the cover. They provide a measure of contamination, a statement that the condition of 'well ventilated' is maintained for applicability of FDS, and an indication of temporal change with which to determine whether or not a steady condition is reached.

In Figure 4-5 the total mass under the cover reduces by about 6% over the 240 s simulation, indicative of a general heating of the space under the cover, due to the fire. There is a similar proportionate reduction in the nitrogen content. As the generation of oxides of nitrogen are not simulated by the model, the change in nitrogen content is entirely a consequence of species conservation, driven by consumption and generation of the other species. A barely noticeable diminution of oxygen takes place too, an indication that combustion in the fire is well ventilated. All three curves in Figure 4-5 are steady by 150 s.

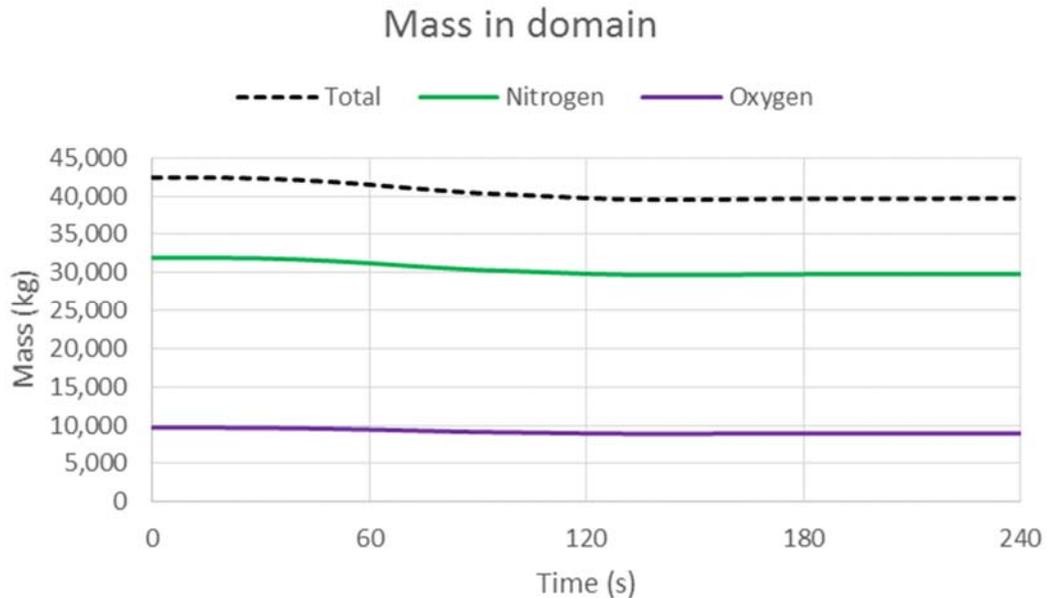


Figure 4-5 Histories of total smoke mass, nitrogen mass, and oxygen mass under the cover.

Figure 4-6 shows reduction in the water vapour content under the cover, a drop in the order of 3%. This implies a small reduction in specific humidity that could impact the perceived comfort of the cover occupants. However, this is not a meaningful factor under fire emergency circumstances. The relatively flat line of the curve from 150 s onward, suggests that the flow condition under the cover is steady.

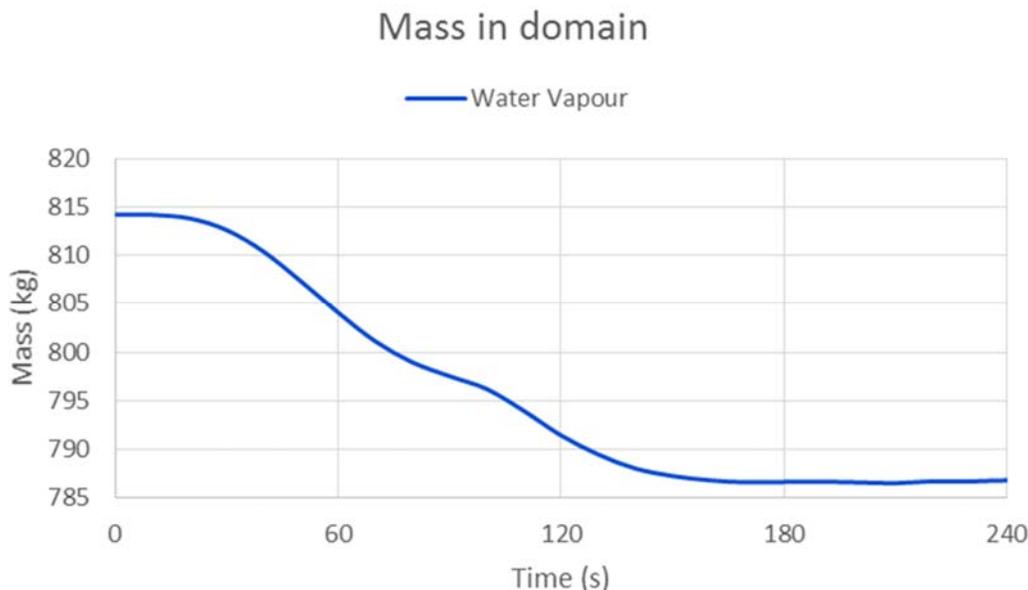


Figure 4-6 History of water vapour mass under the cover.

Figure 4-7 shows the history of carbon dioxide contained under the cover. At the beginning of the simulation the carbon dioxide content is entirely due to its proportion of atmospheric air, 0.0005 kg(CO₂)/kg(smoke). Combustion in the fire generates further carbon dioxide, increasing the total under the cover by about 600%, resulting in a maximum mass fraction of 0.0037 kg(CO₂)/kg(smoke) at about 130 s. Although the fire HRR peaks at 60 s, the carbon dioxide clearly continues to rise until 130 s, followed by a relatively small reduction until it reaches a near steady content by 180 s. The maximum mass fraction of 0.0037 kg(CO₂)/kg(smoke) is the maximum of the average under the cover. In all probability, the actual maximum mass fraction will be greater still.

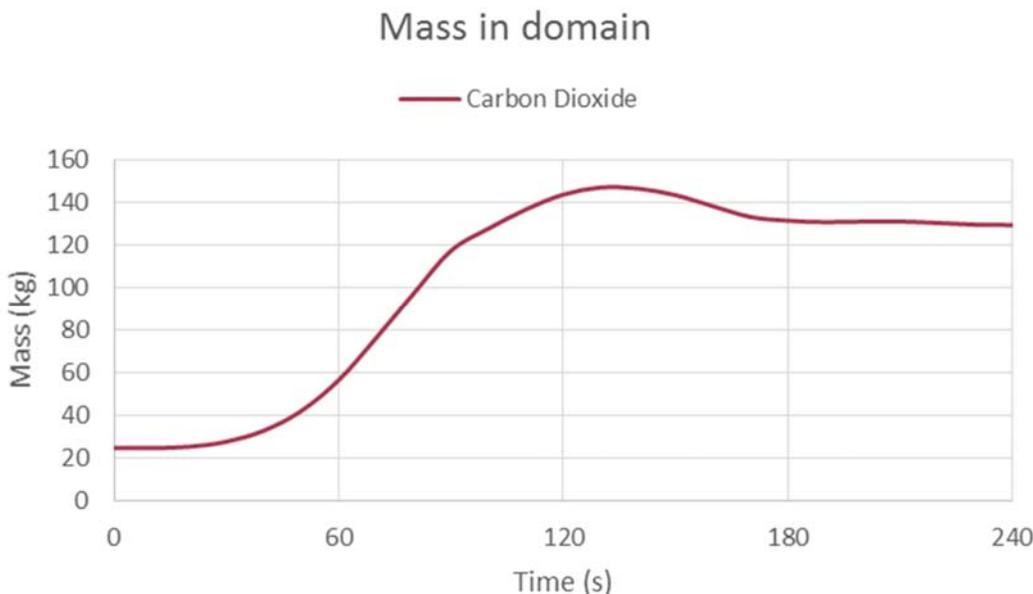


Figure 4-7 History of carbon dioxide mass under the cover.

Figure 4-8 shows the history of soot mass, carbon monoxide mass, and fuel vapour mass, contained under the cover. All three species begin at zero, as carbon monoxide and soot are only products of combustion, and as the fire begins with zero HRR, then the presence of fuel vapour begins as zero also. As the combustion is complete in a well ventilated environment, it stands to reason that the fuel vapour presence will always be small, and only in the immediate proximity of the fire incident. There is no fuel vapour distributed under the cover or that reaches either portal.

The content of both soot (0.000080 kg(soot)/kg(smoke)) and carbon monoxide (0.000039 kg(CO)/kg(smoke)) peak at about 130 s, and subside marginally to near steady values by 180 s. As these values of mass fraction are averaged under the cover, the actual maxima are likely to be higher.

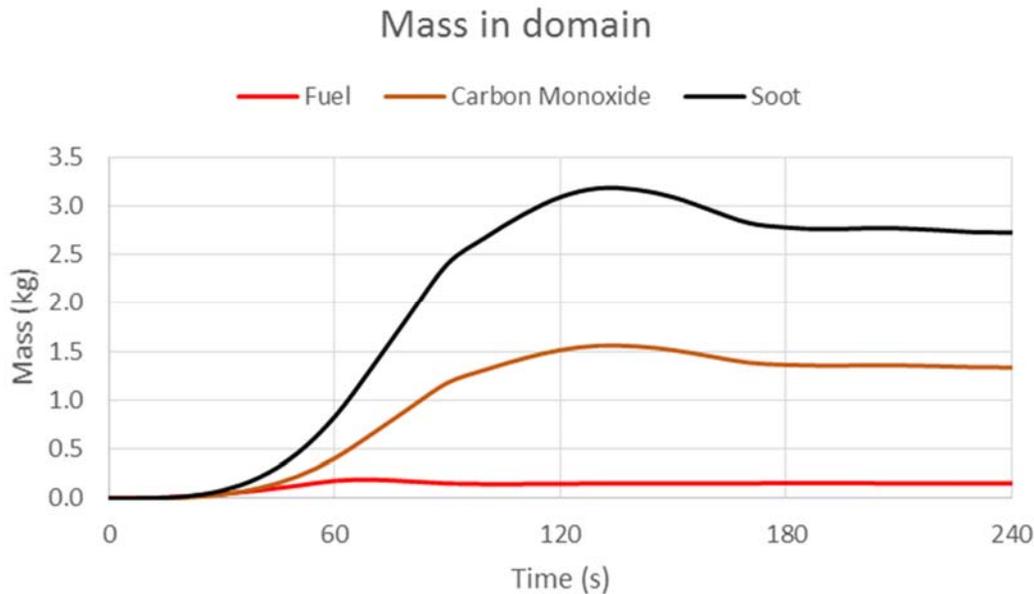


Figure 4-8 Histories of soot mass, carbon monoxide mass, and fuel mass under the cover.

4.3.4. Smoke Isotherms

The dominant force acting on the cover environment, at least until the ventilation has been operating at its maximum for long enough, is buoyancy. By the end of the simulation, the resulting thermal condition under the cover is stable, evidence that a steady thermal condition is reached. The driving force behind buoyancy is temperature difference. To visualise the thermal environment by way of temperature, enables determination of why certain flow phenomena take place.

Figure 4-9 illustrates coloured temperature contours plotted on a longitudinal section through the middle of the cover, passing through the incident vehicle. The range of temperature contours is limited from 60°C up to 520°C, and an envelope described by the 60°C isotherm in purple provides a 3D visual demarcation between hot smoke, and cooler smoke or clearer air.

The first image in Figure 4-9, is an annotated view of the sectioned cover, to explain the oblique viewpoint adopted for the image, in order to prevent the high length to height ratio reducing clarity of the contours and isotherm.

The thermal plume can be seen to rise and impinge on the ceiling above the incident vehicle. The red contours also show how the thermal plume is deflected strongly by the air stream. Downstream contours have reached the exit portal, and the layer depth described by the 60°C isotherm occupies more than half the area of the exit portal.

There is an apparent secondary hotspot downstream of the fire incident, this is a consequence of the wake behind the incident vehicle disrupting local stratification, also evidenced by a small clear patch in the soot images shown in Figure 4-2.

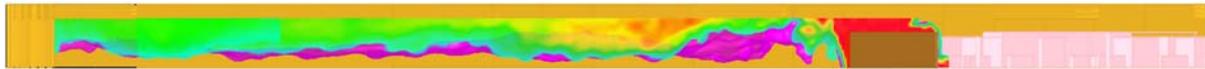
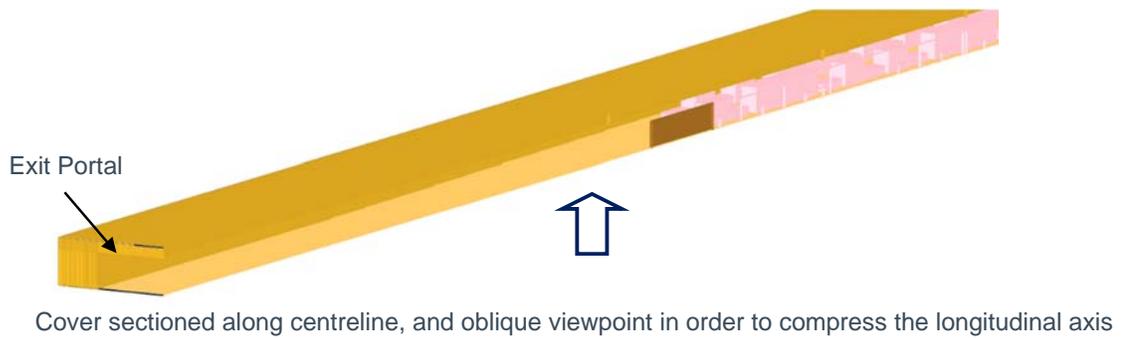


Figure 4-9 60°C isotherm in purple, sliced to show temperature contours above 60°C. Ventilation full capacity, Fire HRR 30 MW.

To gain a more detailed understanding of the thermal evolution of the smoke layer, three isotherms (60°C - green, 200°C - red, and 500°C - purple) are plotted in plan in Figure 4-10. Smoke temperatures immediately above the incident vehicle are shown to be as high as 500°C.

The 200°C isotherm in Figure 4-10 has adopted a 'V' shape, demonstrating the sweeping effect of the ventilation stream. It also occupies the full width of the cover, and extends a small distance downstream of the fire only. This means that vehicles backed-up behind the incident vehicle are not exposed to excessive radiant heat.

The 60°C isotherm in Figure 4-10 occupies the entire cover width from the fire, all the way downstream to the exit portal. This is indicative of the ventilation stream diluting the heat from the fire considerably. However, the thermal environment downstream of the fire remains hostile.

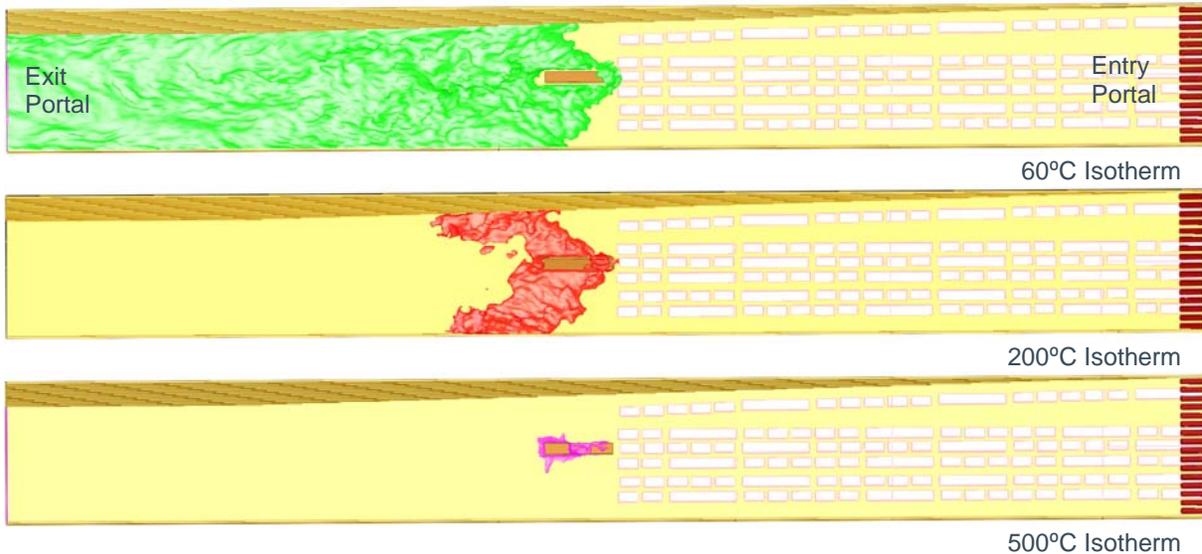


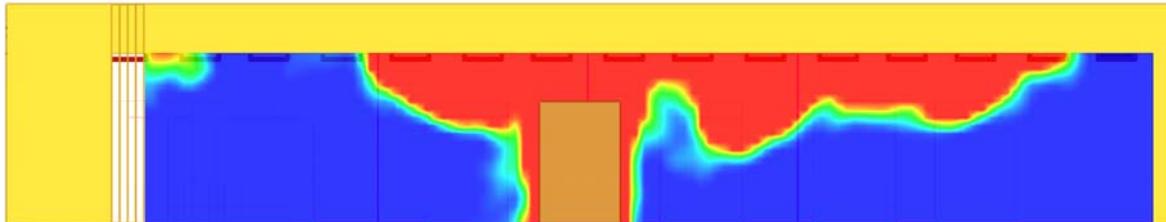
Figure 4-10 Isotherms viewed in plan. Ventilation full capacity, Fire HRR 30 MW.

More information regarding thermal stratification downstream of the fire at the end of the simulation is provided by smoke temperature contours plotted on various cover cross-sections, in Figure 4-11. In the absence of back-layering there is no heating of the cover significantly upstream of the fire, hence no cross-sections are provided for upstream. Note that a restricted temperature scale has been adopted in Figure 4-11, in order to compare like-with-like, and to capture an apparent sharp thermal layer interface, at the largest temperature gradient.

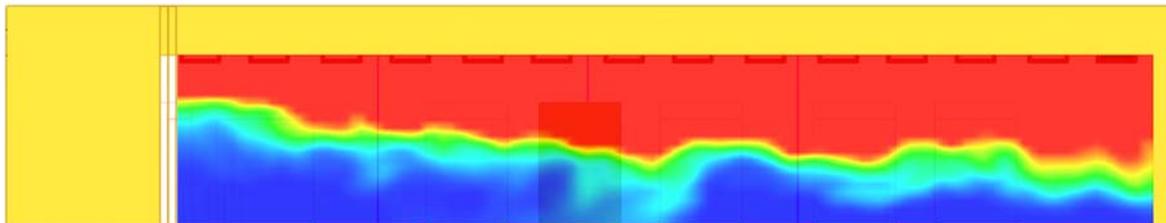
The red contours in Figure 4-11 illustrate temperatures of 90°C and above, they are not intended to indicate a maximum of 90°C. A marked feature of the thermal layer visualised here is that the interface is quite sharp at the fire site. At 200 m (approximately 31 m from the downstream edge of the incident vehicle) the interface

has markedly diffused, and this effect gets more pronounced further downstream, until at the exit portal the interface is considerably smeared. Meaning that thermal stratification at the exit portal is diminished relative to upstream.

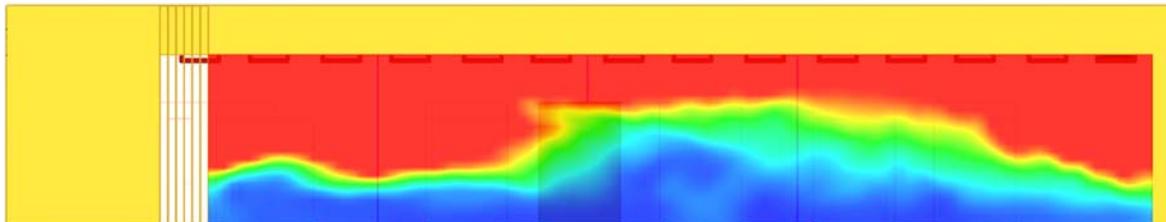
Another marked feature of the thermal layer is undulation in the temperature interface. This is evidence that the LES modeling approach to turbulence used by FDS is resolving local turbulent features. The mean smoke temperature leaving the cover via the exit portal is 56°C, the minimum and maximum values at the exit portal are 41°C and 89°C.



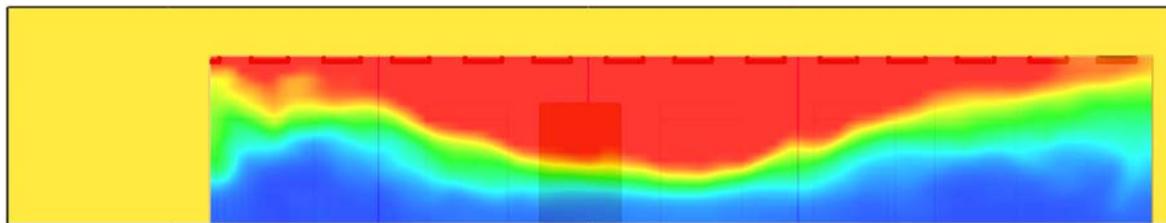
Cross-section through middle of incident vehicle, 160 m from entry portal



Cross-section through cover, 200 m from entry portal



Cross-section through cover, 250 m from entry portal



Cross-section at exit portal, 305 m from entry portal



Figure 4-11 Smoke temperature contours at various cross-sections, on restricted temperature scale.

5. Conclusions

5.1. Comparisons

Comparisons are made in Table 5-1 of the IDA and CFD predicted volume flow rates, and mean temperatures, at the cover portals, induced by the ventilation CRD operating to ventilate the 30 MW mid-cover fire scenario, against a 20 Pa adverse wind pressure at the exit portal.

Table 5-1 Steady portal flow rates and temperatures, determined by IDA (1D) and CFD (3D).

Steady volumetric flow rate and temperature			
Entry (east) Portal		Exit (west) Portal	
IDA (1D)	CFD (3D)	IDA (1D)	CFD (3D)
361 m ³ /s @ 38°C	517 m ³ /s @ 38°C	455 m ³ /s @ 119°C	547 m ³ /s @ 56°C

There are significant differences in the predicted induced ventilation rates between the 1D calculation and the 3D simulation. The CFD simulation has predicted higher flow rates by about 43% at the entry portal, and 20% at the exit portal. There is a commensurate difference in the predicted temperatures for smoke leaving the exit portal, where the higher flow rate predicted by the CFD simulation results in a significantly lower temperature. Analysis of the two methods, to determine what might have contributed to the significant differences, follows:

- Jet fan impulse – a CFD simulation was conducted without a fire, prior to introduction of the fire, in order to determine the jet fan prescription that results in a similar flow rate to that predicted by IDA, also without a fire. See Section 3.3.8.
- Fire – the CFD model incorporates the fire reaction and heat release, so the impedance to ventilation is accounted for rigorously in the CFD simulation. IDA uses a linear relation between the fire HRR and the subsequent impedance to ventilation using a pressure drop factor [1], defaulted to 0.1 Pa/MW but set in [1] to 0.2 Pa/MW for this study.
- Grade – in this study the CFD model incorporates the road/cover grade by way of adjusting the gravity vector. IDA incorporates the stack effect.
- Wind pressure – the CFD model incorporates the adverse wind at the exit portal by applying the dynamic wind pressure excess relative to the entry portal pressure. IDA includes allowance for the adverse wind pressure.
- Wall friction – in this study the CFD model incorporates wall friction using the smooth wall assumption. IDA includes wall friction. However, for the reason reported in Section 3.3.8, the influence of wall friction is accounted for by establishing a CFD solution for the jet fans representation, mimicking an IDA solution without a fire.
- Wall heat transfer – in this study the CFD model incorporates walls, ceiling and the road, at a constant surface temperature set to the initial cover temperature of 38°C. IDA incorporates heat loss to the walls that allows the wall temperature to vary.
- Incoming air temperature – in this study the incoming air temperature was set to 38°C CFD as this represents the most challenging condition for tunnel ventilation performance assessment.
- Stationary traffic – the CFD model incorporates stationary traffic by explicit partial blocking of the cover. IDA incorporates the blockage effect of stationary traffic.

5.2. Ventilation CRD assessment

Generally, the range of output generated by the CFD simulation provides for a comprehensive demonstration of the ventilation performance. The CFD results demonstrate that if used in a timely manner, the ventilation

CRD will prevent back-layering of smoke and hot gases, thus protecting vehicles and their occupants backed-up behind the incident vehicle, and attendant emergency services, from the harmful effects of fire. In this respect, the CFD model supports the CV calculation used by IDA.

6. References

- [1] Ventilation and Fire Life Safety Report.
I-70 East Project, Denver, CO. Partial Covered Lowered Alternative Atkins Report Number 5115267/REPORT/003, September 2015.
- [2] Fire Dynamics Simulator Users Guide. NIST Special Publication 1019, Sixth Edition.
<https://code.google.com/p/fds-smv/>
- [3] Fire Dynamics Simulator Technical Reference Guide, Volume 1: Mathematical Model.
NIST Special Publication 1018, Sixth Edition. <https://code.google.com/p/fds-smv/>
- [4] Fire Dynamics Simulator Technical Reference Guide, Volume 2: Verification.
NIST Special Publication 1018, Sixth Edition. <https://code.google.com/p/fds-smv/>
- [5] Fire Dynamics Simulator Technical Reference Guide, Volume 3: Validation.
NIST Special Publication 1018, Sixth Edition. <https://code.google.com/p/fds-smv/>
- [6] Smokeview, a Tool for Visualising Fire Dynamics Simulation Data, Volume 1: Users Guide.
NIST Special Publication 1017-1, Sixth Edition. <https://code.google.com/p/fds-smv/>
- [7] Smokeview, a Tool for Visualising Fire Dynamics Simulation Data,
Volume 2: Technical Reference Guide. NIST Special Publication 1017-2, Sixth Edition.
<https://code.google.com/p/fds-smv/>
- [8] Smokeview, a Tool for Visualising Fire Dynamics Simulation Data,
Volume 3: Verification Guide. NIST Special Publication 1017-3, Sixth Edition.
<https://code.google.com/p/fds-smv/>
- [9] Reference drawing: 19631DES_Cover Typical Section for UK.dgn I-70 mainline typical section cover.
12th February 2014.
- [10] Preliminary reference drawings: 13599DES_Procurement_Prfl10_Lowered_I70.dgn;
13599DES_Procurement_Prof53/54/55/56_Lowered_Steele WB On.dgn. 12th December 2014
- [11] Preliminary reference drawings: 13599DES_Procurement_Plan10_Lowered_I70 Mainline.dgn;
13599DES_Procurement_Plan53/54_Lowered_Steele WB On.dgn;
13599DES_Procurement_Plan55/56_Lowered_Steele EB Off.dgn . 12th December 2014
- [12] PCL-ML 'Full-Build', Link Source Type data file for MOVES modeling, supplied by ANA. Email from Daniel Liddle to Simon Drake, Gary Clark, cc. Andrew Iltis. FW: I70 East Covered Roadway - MOVES Modelling Sent on 6th February 2015 18:15 (GMT time).



© Atkins Ltd except where stated otherwise.

The Atkins logo, 'Carbon Critical Design' and the strapline 'Plan Design Enable' are trademarks of Atkins Ltd.