CFD MODEL FOR TRANSVERSE VENTILATION SYSTEMS

Sam S. Levy, Jason R. Sandzimier, Norris A. Harvey, Elana M. Rosenbluth
Parsons Brinckerhoff
One Penn Plaza
New York, NY 10119 USA

Kailash C. Karki, Suhas V. Patankar
Innovative Research, Inc.
3025 Harbor Lane N., Suite 300
Plymouth, MN 55447 USA

published in

Proceedings of the First International Conference on
Tunnel Fires and One Day Seminar on Escape from Tunnels, pp. 223–233
5–7 May 1999, Lyon, France
Organized and sponsored by Independent Technical Conferences Ltd.

ABSTRACT

During Phase IV of the Memorial Tunnel Fire Ventilation Test Program, a general Computational Fluid Dynamics (CFD) code was customized and validated specifically for tunnel application. For transverse ventilation, a novel approach was developed for modeling the interaction between ventilation ducts and the tunnel. A network model, comprised of nodes and links, is used to represent the ducts while a field model is used to represent the tunnel. These models interact with each other through boundary conditions. The paper presents the details of the network model, the method for integrating this model with the basic field model, and the overall solution procedure. The general application of the CFD model to the Memorial Tunnel is discussed. Model predictions are compared with test data from selected fire tests for both steady-state and transient conditions.

1.0 INTRODUCTION

The Memorial Tunnel Fire Ventilation Test Program (MTFVTP) evolved from the need to better understand the capabilities of tunnel ventilation systems during a fire emergency. The Memorial Tunnel, an abandoned road tunnel in West Virginia, was modified, retrofitted with new ventilation equipment, and instrumented to evaluate ventilation system performance during full-scale testing as a function of system type and capacity, and fire size. Ventilation systems tested included longitudinal ventilation using jet fans, natural ventilation, full transverse ventilation, partial transverse ventilation, and partial transverse ventilation supplemented with special extraction techniques. A total of 98 full-scale tests were carried out with fires ranging in intensity from 10 to 100 MW.

The test program comprised four phases of work. The first three phases addressed test program development, test facility design and construction, testing and data evaluation. Phase IV focused on development and validation of a customized Computational Fluid Dynamics (CFD) code specifically for tunnel application. Data from the full-scale fire tests was used as the basis for validation.

The CFD model is geared towards individuals concerned with fire/life safety in tunnels from a perspective of analysis, design, and operation of ventilation systems. The primary objective established for the model is the ability to simulate the interactive effects of a tunnel fire and the ventilation system to determine the unsafe regions of the tunnel, that is, the regions where the hazardous effects of the fire (smoke and high temperature) are confined, and how these regions are affected by the ventilation system configuration, capacity, and operation.

The customized CFD code is based on an existing general-purpose computer program for the analysis of fluid flow, heat transfer, and related processes (IRI, 1996). The customization work included development, implementation, and validation of sub-models to address certain features required for the tunnel model.

One such feature is the ability to model tunnel ventilation air ducts to address transverse ventilation systems. The requirements for this feature were many. The model had to be sufficiently flexible to address not only the ventilation configurations tested in the Memorial Tunnel but also possible extensions, variations, and combinations of these systems. The model had to be able to deliver each duct system's flow capacity and distribution, and adjust them accordingly to account for the effects of a tunnel fire. The exchange of flow, heat, and smoke at the tunnel/duct wall interface had to be accounted for properly. In addition, this had to be accomplished without overburdening the computational process. To meet these requirements, a novel approach was developed.

2.0 DETAILS OF THE MATHEMATICAL MODEL

In transverse ventilation systems, air is supplied to and exhausted from the tunnel through ventilation ducts. The supply and exhaust rates through the ventilation ducts are not known *a priori*; these depend not only on the characteristics of the duct systems but also on the conditions within the tunnel. A complete model for such systems, therefore, requires a method for calculating the fluid flow and heat transfer within the tunnel, a method for calculating the supply and exhaust rates for the ducts, and a procedure for interacting the two.

A field model based on CFD is used to calculate the flow and heat transfer characteristics within the tunnel. A flow network model is used to calculate the flow and heat transfer through the ventilation ducts. A special procedure has been developed to couple these models so that they fully interact with each other. The specific details of these three components of the overall model for transverse ventilation systems are presented in the following sections.

2.1 Field Model for Tunnel (Tunnel Model)

The tunnel model uses a numerical method to solve the three-dimensional, time-dependent equations (field equations) describing the laws of conservation for mass, momentum, energy, turbulence parameters, and species, subject to the given set of boundary conditions. It is based on the buoyancy-augmented k-ε turbulence model (Cox, 1995) and includes component models for representing fire, radiation heat transfer from fire, smoke movement, and wall roughness.

2.1.1 Governing Equations

The governing equation for the transport of mass, momentum, energy, turbulence parameters and other quantities can be cast, using the Cartesian tensor notation, in the following general form (Patankar, 1980):

$$\frac{\partial}{\partial t}(\rho\phi)+\frac{\partial}{\partial x_i}(\rho u_i\phi)=\frac{\partial}{\partial x_i}\left(\Gamma\frac{\partial\phi}{\partial x_i}\right)+S \tag{1}$$

where ϕ is the general dependent variable, ρ is the fluid density, Γ is the generalized diffusion coefficient, and S is the source term. The density is calculated from the perfect gas law. The expressions for the diffusion coefficients and the source terms appearing in the transport equations are well known (see, for example, Cox, 1995) and are not presented here.

2.1.2 Boundary Conditions

Tunnel Portals. The tunnel portals can be specified as inflow/outflow boundaries or as "free" boundaries with known values of pressure, depending on the physical situation being modeled. At an inflow boundary, values of all variables are specified. At an outflow boundary, the diffusion flux normal to the boundary is assumed to be zero and no other information is needed. At a free boundary, the value of pressure is specified. The given value of pressure is interpreted as total pressure at the inflow points and static pressure at the outlet points.

Tunnel Walls. At a solid-fluid interface, the wall-function approach (Launder and Spalding, 1974) is used. The approach outlined by Jayatilleke (1969) is followed to account for the influence of wall roughness on the standard wall functions.

2.1.3 Solution Procedure

The governing equations for the tunnel model are solved using the finite-volume method described by Patankar (1980). The implicit differencing scheme is used for the unsteady term in the equations. The convection-diffusion fluxes are approximated using the Power-law scheme. The coupling between the velocity and pressure fields is handled using the SIMPLER algorithm. The algebraic equations are solved using the TriDiagonal-Matrix Algorithm (TDMA).

2.1.4 Representation of Fire

The fire is represented as a source of heat and mass. The heat release rate due to combustion is prescribed as a volumetric heat source in a *postulated* fire region. The model needs information on the flame size and shape and the volumetric heat release rate and its distribution. The heat release rate is computed from the rate of fuel consumption ($\dot{m}_{fu}$), the heating value of the fuel (H_{fu}), and the combustion efficiency (η), as

$$Q=\dot{m}_{fu}H_{fu}\eta \tag{2}$$

In the fire region, the energy equation includes an additional source term, which is calculated on a unit-volume basis as

$$S_{h,fire}=\frac{Q}{V_{fire}}(1-\chi_R) \tag{3}$$

where V_{fire} is the volume of the fire region and χ_R is the fraction of the total heat released from the fire that is lost to the tunnel walls by radiation, without influencing the temperature distribution within the tunnel.

2.1.5 Representation of Smoke

In the tunnel model, a separate conservation equation is solved for smoke. This equation contains a source term in the fire region where the combustion process takes place. The total rate of smoke production is calculated from the rate of fuel consumption and the stoichiometric ratio for the fuel, assuming complete combustion. On a unit-volume basis, this source term is calculated as

$$S_{smoke} = \frac{\dot{m}_{fu}(1+s)}{V_{fire}} \tag{4}$$

where s is the stoichiometric ratio (kg of air / kg of fuel) for the fuel.

2.2 Flow Network Model for the Ventilation Ducts

A ventilation duct system is represented as a network of links and nodes. The values of pressure, temperature, and smoke concentration are stored at the nodes. At boundary nodes, the values of these variables are known. At the remaining nodes, these values are unknown and are calculated by the network model. Each link in the network represents a fluid path and is associated with an upstream and a downstream node. A link is characterized by the aerodynamic resistance, the flow rate, and the heat transfer coefficient. The details of the network model are presented.

2.2.1 Governing Equations

The basic governing equations in a flow network model are the mass continuity equation at a node, the correct relationship between the pressure drop and the flow rate (momentum equation) for a link, and the energy equation at a node.

The mass continuity equation for node *i* can be expressed as

$$\sum_{j=1}^{J} \left(\rho_{ij} Q_{ij}\right) = -m_i \tag{5}$$

J is the total number of links associated with node *i*. Here, ρ is the density, *Q* is the volumetric flow rate, and m is the external mass flow into node *i*. In Eq. (5) and subsequent equations, the subscript *i* denotes the value of the quantity at node *i* and the subscript *ij* indicates reference to the *j*th link connected to the node *i*.

The pressure drop-flow rate relationship for a link is expressed as

$$\Delta p_{ij} = L(Q_{ij}) + F(Q_{ij}) \tag{6}$$

where *L* is the flow resistance (frictional loss and minor losses) in the link and *F* represents additional driving forces such as pressure rise due to a fan.

The energy equation for a node is given by

$$\sum_{j=1}^{J} \left(\rho_{ij} Q_{ij} H_{ij}\right) = S_i^H \tag{7}$$

where *H* is the enthalpy and S_i^H is the total energy loss from the various links connected to the node *i* that is attributed to node *i*.

2.2.2 Link Characteristics

To complete the problem specification, additional expressions are required for the flow resistance and energy loss from a link. The frictional loss is expressed in terms of an aerodynamic resistance coefficient *R* as

$$L(Q_{ij}) = R\left|Q_{ij}\right|Q_{ij} \tag{8}$$

The heat loss from a link is calculated as

$$q_{ij} = h_{ij} A_{ij} (T_f - T_w) \tag{9}$$

where h is the heat transfer coefficient, *A* is the surface area, T_w is the wall temperature for the link, and T_f is the fluid temperature in the link. The heat transfer coefficient is calculated from the mass flow rate in the link and accounts for the duct wall roughness (see, for example, Burmeister, 1983).

2.2.3 Fan Pressure Rise

The pressure rise due to a fan is expressed as a function of the volumetric flow rate *Q* as

$$\Delta p_f = a + bQ + cQ^2 + dQ^3 \tag{10}$$

where the coefficients *a*, *b*, *c* and *d* are determined from four points on the fan performance curve.

2.2.4 Boundary Conditions

The values of pressure and temperature (or enthalpy) are specified at the boundary nodes in the network. These boundary values determine the values of pressure and temperature at the remaining nodes and the flow rates for the links.

2.2.5 Solution Procedure

The governing equations for the flow network model are discretized using the well-established procedure described by Patankar (1980). The SIMPLE algorithm is used to handle implicitly the relationship between the flow rate and the node pressures. A matrix inversion technique is used to solve the final algebraic equations.

2.3 Interaction Between the Tunnel and Duct Network Models

As stated earlier, the tunnel model and the duct model are coupled with each other via the supply and exhaust rates. These rates depend on the conditions within the tunnel as well as on the characteristics of the duct system. In the overall model, these two component models interact with each other through boundary conditions in the following manner:

- The network model provides the air supply/extraction rates for the tunnel model. These ventilation rates are used as boundary conditions (sources and sinks) in the tunnel model.

- The tunnel model provides the values of pressure and temperature at selected nodes (to be defined later) in the duct network representation. These boundary conditions in conjunction with the conditions prevailing in the ducts determine the values of pressure, temperature, and mass flow rates throughout the duct network.

2.3.1 Representation of Tunnel Conditions and Tunnel-to-Duct Communication

The methodology developed to model the interaction between the tunnel and the ventilation ducts reflects the exchange of flow that occurs through slots (ports and flues) on the tunnel walls or ceiling. It should be noted that the dimensions of the slots might be much smaller compared to the grid spacing used in the field model for the tunnel. As a result, it is generally not possible to represent slots as individual links in the network model.

The approach followed in the model is illustrated via the sample network for an exhaust duct system shown in Figure 1. For each ventilation duct, the tunnel adjacent to the duct is divided into a number of zones (*tunnel zones*) in the longitudinal and the width or height (depending on the location of the duct) directions. A tunnel zone may include a number of control volumes of the tunnel model. Further, the zone boundary along the tunnel wall where the duct is located may span over a number of slots. The conditions within a tunnel zone are represented by average values of pressure and temperature in the zone. These values are calculated from the solution provided by the tunnel model. These zones are represented as boundary nodes (or *tunnel* nodes) in the network model. For each boundary node in the tunnel, a corresponding node (*duct* node) is introduced in the ventilation duct, and each set of the tunnel and duct nodes is connected by a special link (*tunnel-to-duct* link), which represents the effective slot area associated with the zone. These tunnel nodes and tunnel-to-duct links allow interaction between the tunnel and the duct.

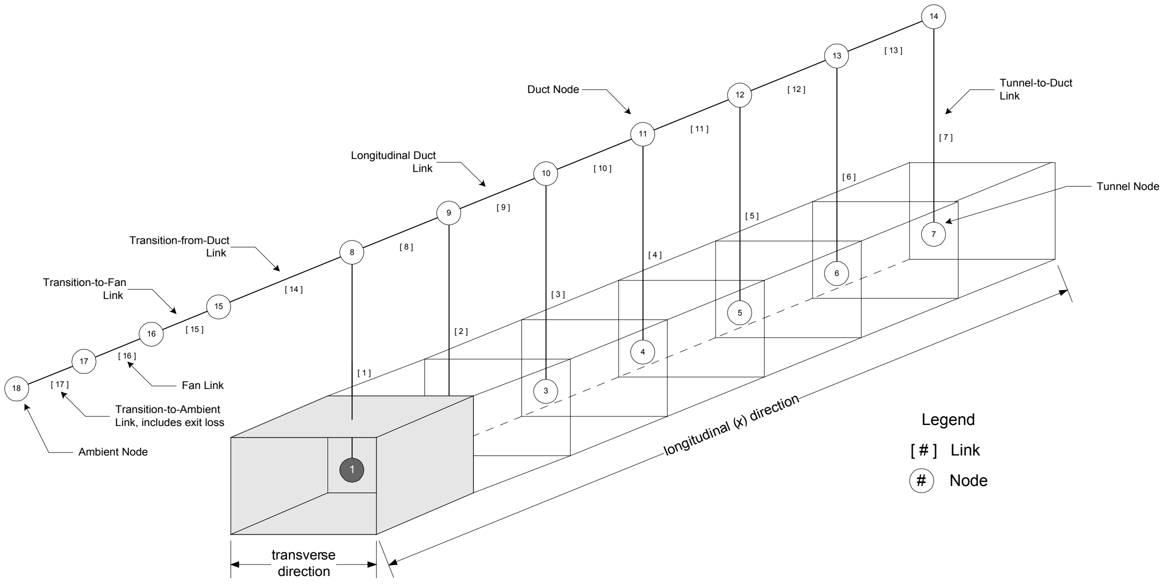


Figure 1: Example of an Exhaust Duct Network

2.3.2 Overall Solution Procedure

The coupling between the tunnel and duct network models is handled iteratively until a converged solution is obtained. The steps in the overall solution procedure are as follows:

1. Select a suitable grid for the tunnel (CFD) model.
2. Create a node-link representation of the duct with special links to the tunnel nodes.
3. Provide a guess for the conditions within the tunnel.
4. Calculate the average pressure and temperature for the tunnel nodes. The calculation of average pressure includes all the control volumes that form the given tunnel zone. The calculation of temperature, however, includes only the control volumes adjacent to the tunnel boundary where the duct is located.
5. Solve the network model to obtain air supply/extraction rates.
6. Solve the tunnel model with the supply/extraction rates specified as mass sources/sinks in the control volumes adjacent to the tunnel boundary where the duct is located.
7. Repeat steps 4 through 6 until convergence is achieved.

In the implementation of the network model, fluid and thermal inertia terms are not included in the governing equations. Thus a steady form of the equations is used even in a transient simulation, and the resulting output from the network model represents the steady-state conditions corresponding to the tunnel conditions at a given time. To introduce the fluid inertia effect indirectly, a run-up or run-down feature is implemented in the fan operation. In this feature, the fan pressure rise is made a function of time according to a prescribed relationship.

3.0 MODEL VALIDATION

As part of the validation effort in Phase IV, 10 fire tests, representing four transverse ventilation configurations, were modeled and studied extensively. The success of this work demonstrated the flexibility of the transverse model to address multiple ventilation system configurations. To demonstrate the effectiveness of the modeling approach, results from simulations of a two-zone partial transverse ventilation fire test are presented.

3.1 Tunnel Configuration

The Memorial Tunnel is about 853 m long and has a constant 3.2 percent grade from south-to-north. During transverse ventilation testing, the ceiling above the roadway served as the floor of the duct system which extended essentially the length of the tunnel. A vertical partition divided the duct into supply and exhaust sections, each served by fans located in equipment rooms at the south and north ends of the tunnel, respectively. The variable speed drive fan systems were reversible, enabling multiple ventilation system configurations and capacities. The unobstructed rectangular tunnel cross-sectional area was 8.74 m wide by 4.33 m tall.

The tunnel was extensively instrumented with thermocouples, bi-directional pitot tubes, and video cameras. The fires were generated by burning controlled amounts of No. 2 distillate fuel oil in steel fire pans located approximately 240 m from the south portal. The fuel oil was pumped to the pans from a remote location. Continuous measurements of the fuel flow rate, fire pan weight, and combustion product gas concentrations were used to compute the fire heat release rate. The combustion efficiency of the fire was estimated from CO and CO_2 gas product concentrations sampled at the ceiling level, approximately 60 m away from the fire.

During two-zone partial transverse ventilation testing, the tunnel was divided into two ventilation zones by placing a bulkhead at the midpoint of the duct. Air was supplied to the northern half of the tunnel and exhausted through the southern half. The tunnel and the ventilation configuration are depicted in Figure 2.

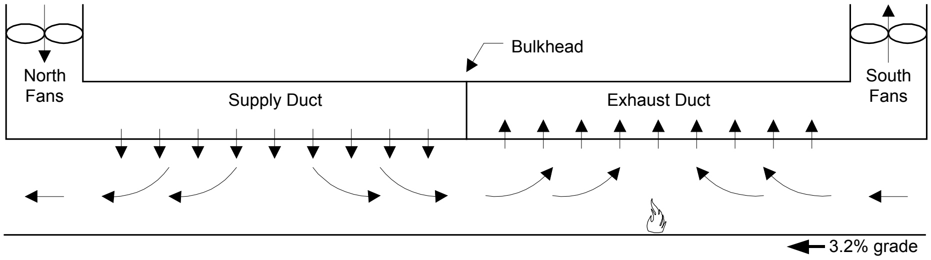


Figure 2: Two-Zone Partial Transverse Ventilation

3.2 Tunnel and Duct Numerical Representation

The tunnel is represented in the CFD field model by a structured Cartesian coordinate, non-uniform grid consisting of 29 cells in the width direction, 14 cells in the height direction, and 369 cells in the longitudinal direction. The data measurement and collection equipment used during the testing created a significant amount of flow resistance within the tunnel. Solid objects representing the various components of the instrumentation were incorporated in the model to represent their resistive effects. The tunnel portals are modeled as pressure boundaries. With the exception of the walls in the fire zone, a 120-m long section surrounding the fire, the wall boundaries were modeled as rough concrete, isothermal surfaces. For testing, the tunnel walls and ceiling in the fire zone were sprayed with several centimeters of insulation as a protective measure. The boundary surfaces in this region were modeled as adiabatic.

For the network representation, the tunnel under the supply and exhaust ducts was divided into tunnel zones approximately 15 m long, spanning the 3-m width of the ventilation slots. For a given fan flow and duct flow distribution, the resistances of the various network links were determined from duct pressure distribution curves (Singstad, 1929). The fan operating pressure was determined from the total pressure loss in the duct system. Using the Memorial Tunnel fan performance curves and the fan laws, a characteristic fan performance curve was determined and used as input for the network model.

3.3 Validation Effort

The length and quality of data collection during test 251B, a nominal 20 MW fire test, presented opportunities to perform both steady-state and transient simulations. To determine the impact of a delayed ventilation response, the fan system was not activated until 2 minutes after the fire was fully developed. During this period, smoke propagated 300 m north and 60 m south of the fire. Shortly after the ventilation system was initiated (132 m^3/s supply and 132 m^3/s exhaust), the tunnel conditions steadied and the smoke and hot gases were contained within 75 m of the fire.

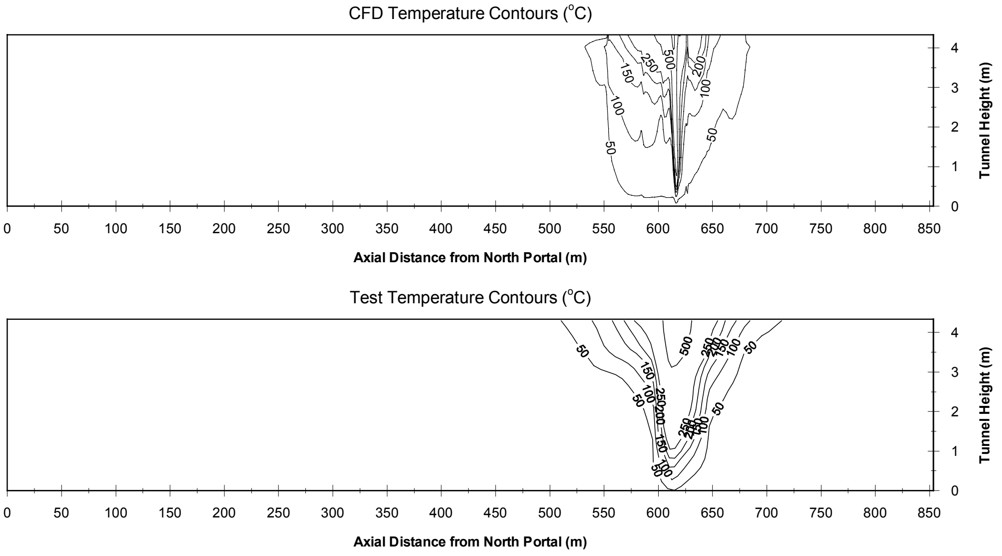
The steady-state simulation was meant to reproduce the quasi-steady conditions observed after the fan system was activated. The simulated convective fire heat release rate, 12.0 MW, was determined from fuel consumption data, combustion efficiency measurements, and 0.3 for the radiative fraction.

The transient simulation models a 12-minute time period beginning at the start of the test. The fire heat release rate linearly increased from 0 to 12 MW during the first 160 seconds of the simulation and was then held constant for the duration. The supply and exhaust fans were initiated at 125 seconds and the fan run-up time was 65 seconds. Five-second time steps were used throughout the transient simulation.

Comparisons of simulation results with measured test data are presented in the following graphics. Results from the steady-state simulation are presented first (see Figure 3), followed by results for the transient simulation (see Figures 4 and 5).

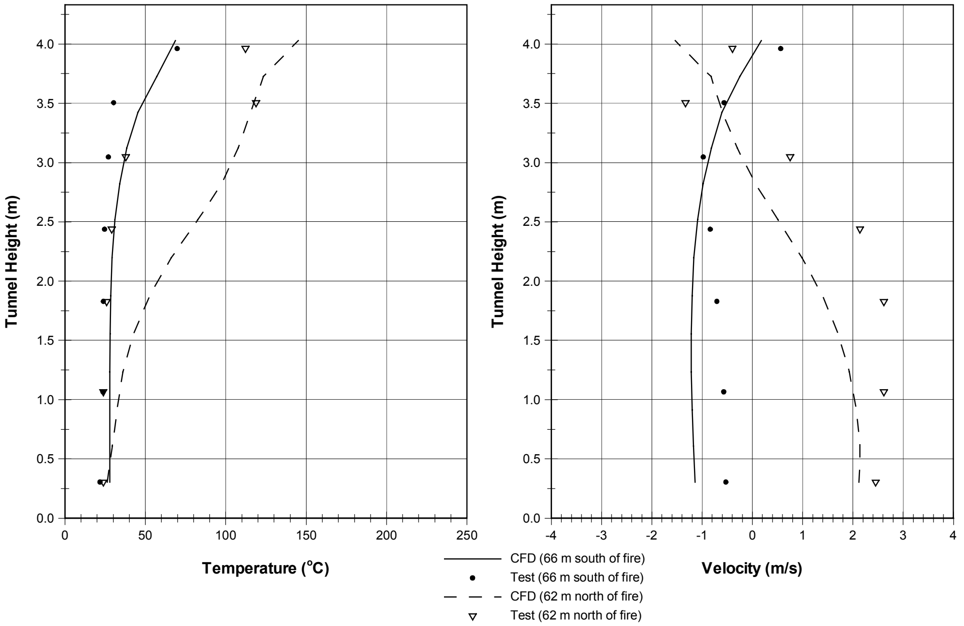
Steady-State Analysis. The temperature contours and the bulk temperature comparisons show that the model predicts the extent of the hazardous region very well. The temperature and velocity profiles also compare reasonably well with the test data. Furthermore, the bulk flow plot shows that the flow distribution in the tunnel is well-predicted.

Transient Analysis. Temperature results at three points in time are presented and flow results during the entire simulation/test is shown for two select tunnel locations. Note that in these plots, north-to-south flow is positive. During the natural ventilation period, a similar level of temperature stratification is predicted, although the extent of high temperature propagation is slightly under-predicted. This difference may be attributed to the small amount of south-to-north ambient flow during the test that was ignored in the simulation and to uncertainty in modeling the fire growth during the first 160 seconds. After the fans are started, the rate of temperature recovery in the test is closely reflected in the simulation. By the end of the simulation, predicted and measured conditions correlate quite well.

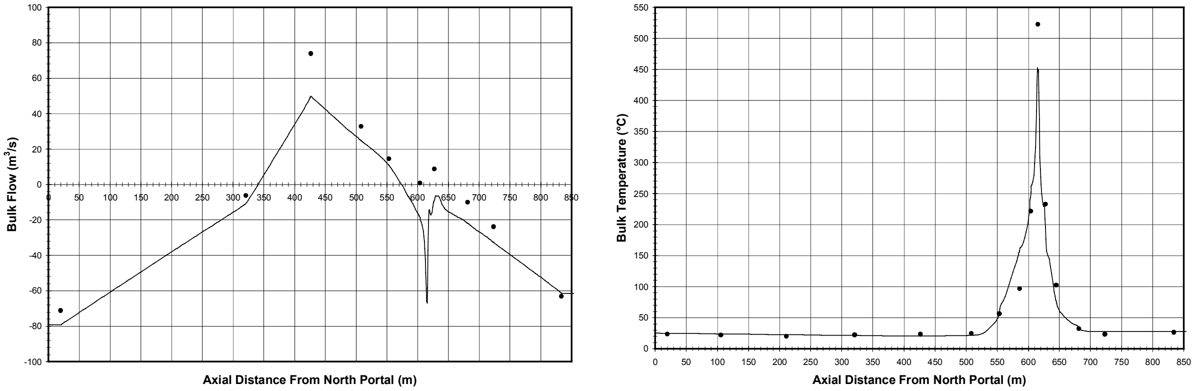


Temperature Contours Along Center of Tunnel Roadway

Figure 3: Steady-State Simulation Results

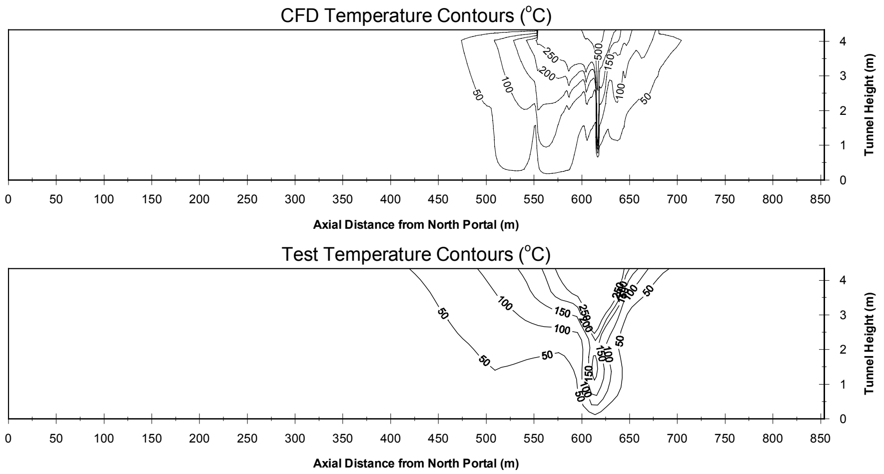


Temperature and Velocity Centerline Profiles



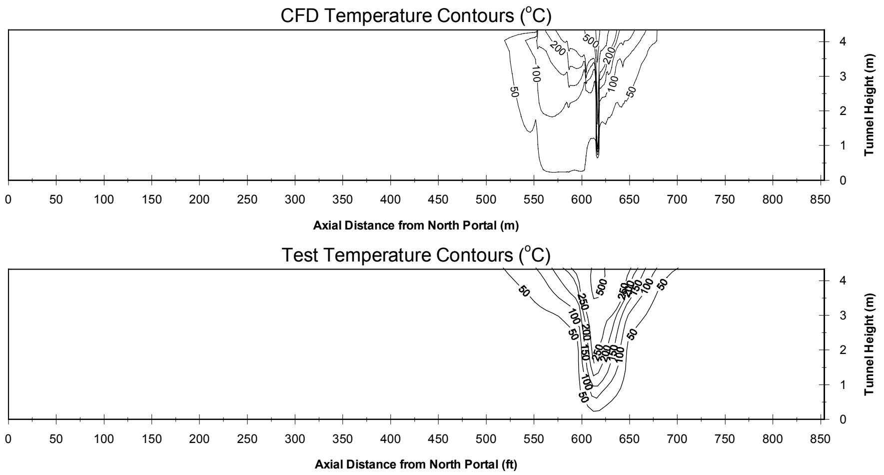
Bulk Flow and Bulk Temperature Distribution Throughout the Tunnel

Figure 3 (*cont.*): Steady-State Simulation Results


CFD Temperature Contours (^{o}C)

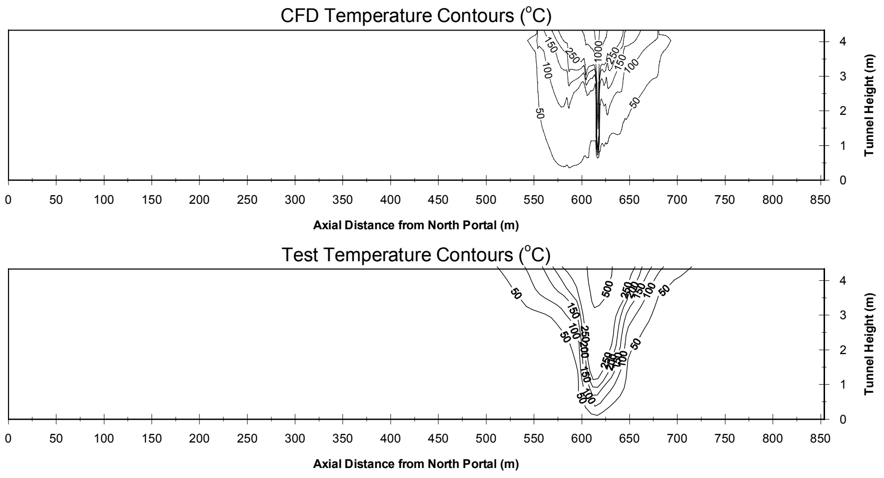
Test Temperature Contours (^{o}C)

Elapsed Time 120 Seconds


CFD Temperature Contours (^{o}C)

Test Temperature Contours (^{o}C)

Elapsed Time 305 Seconds


CFD Temperature Contours (^{o}C)

Test Temperature Contours (^{o}C)

Elapsed Time 730 Seconds

Figure 4: Transient Temperature Contour Results

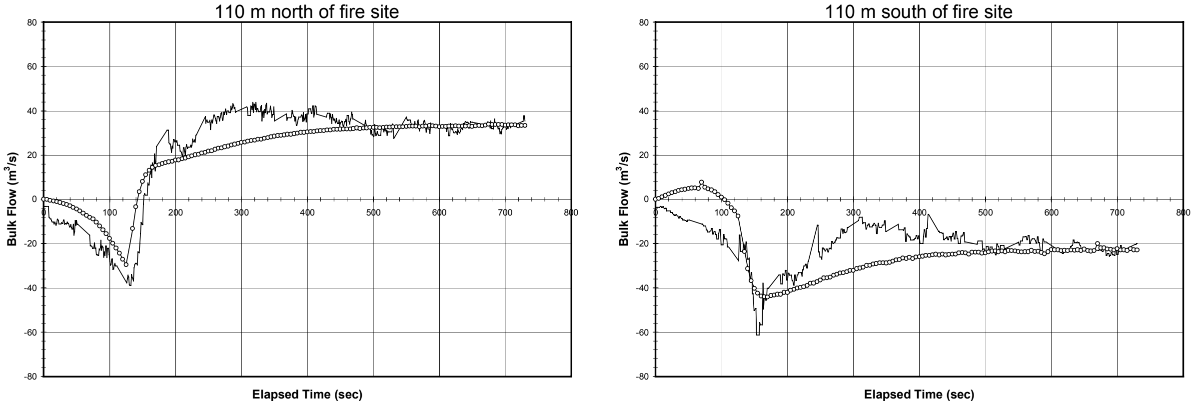


Figure 5: Transient Bulk Flow Results

4.0 CONCLUSIONS

In the development and validation of a CFD model specifically for tunnel ventilation, a network model was successfully used to simulate the interaction between tunnel duct systems and conditions within the tunnel. In particular, the network model is: 1) sufficiently flexible to simulate multiple duct system configurations; 2) able to deliver the desired duct capacity and distribution and adjust them accordingly to account for the effects of a fire; and 3) able to properly account for the exchange of flow, heat, and smoke at the tunnel/duct wall interface.

References Cited

Burmeister, L.C. 1983. *Convective Heat Transfer*. John Wiley and Sons.

Cox, G., ed. 1995. *Combustion Fundamentals of Fire*. Academic Press.

Innovative Research, Inc. 1996. COMPACT-3D Version 4.0. Innovative Research, Inc.

Jayatilleke, C.L.V. 1969. The Influence of Prandtl Number and Surface Roughness on the Resistance of Laminar Sub-layer to Momentum and Heat Transfer. *Progress in Heat and Mass Transfer*, Vol. 1, pp. 193–329. Pergamon Press.

Launder, B.E., and D.B. Spalding. 1974. The Numerical Computation of Turbulent Flows. *Computer Methods in Applied Mechanics and Engineering*, Vol. 3, pp. 269–289.

Patankar, S.V. 1980. *Numerical Heat Transfer and Fluid Flow*. Taylor and Francis.

Singstad, O. 1929. Ventilation of Vehicular Tunnels. World Engineering Congress.