

5.0 CFD Model for Tunnel Ventilation

This chapter provides details of the tunnel fire ventilation model developed under the Memorial Tunnel Fire Ventilation Test Program (MTFVTP), Phase IV. The model belongs to the category of “field models.” The model solves, using a numerical method, the three-dimensional, time-dependent equations (field equations) describing the laws of conservation for mass, momentum, energy, turbulence parameters, and species, subject to the specific boundary conditions of the problem, throughout the domain of interest.

The tunnel ventilation model is based on the buoyancy-augmented $k-\varepsilon$ turbulence model and includes component models for representing jet fans, the fire region, radiation heat transfer from the fire, smoke movement, conjugate heat transfer and wall roughness. For transverse ventilation systems, the ventilation ducts are represented by a flow network which interacts with the tunnel. The model is based on COMPACT-3D, a general-purpose computer program for the analysis of fluid flow, heat transfer, and related processes. The details of the basic model and the individual component models are presented. A brief description of the numerical method used is also presented. Additional details are available in the Reference Manual for COMPACT-3D [1] and Patankar [2].

5.1 Description of the Basic Model

5.1.1 General Form of the 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 [2]:

$$\frac{\partial}{\partial t}(\rho\phi) + \frac{\partial}{\partial x_i}(\rho u_i \phi) = \frac{\partial}{\partial x_i} \left(\Gamma \frac{\partial \phi}{\partial x_i} \right) + S \quad (5-1)$$

where ϕ is the general dependent variable, ρ is the air density, Γ is the generalized diffusion coefficient, and S is the source term. Note that for turbulent flows, ϕ represents the time-averaged quantity. The density is calculated from the perfect gas law. The standard expressions for the diffusion coefficients and the source terms appearing in different transport equations are given in Table 5.1-1. The component models for the fire, radiation, jet fans, and the ventilation ducts introduce additional source terms in these equations. These details are presented in the appropriate sections related to the component models.

Table 5.1-1 Diffusion Coefficients and Source Terms for Conservation Equations

Equation	ϕ	Γ	S
Mass	1	–	0
Momentum	u_i	$\mu + \mu_t$	$-\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu_t \frac{\partial u_j}{\partial x_i} \right) + \rho g_i - \frac{2}{3} \rho \frac{\partial k}{\partial x_i}$
Kinetic energy	k	$\mu + \mu_t / \sigma_k$	$P + G_B - \rho \varepsilon$
Rate of dissipation	ε	$\mu + \mu_t / \sigma_\varepsilon$	$[c_1(P + G_B) - c_2 \rho \varepsilon] \frac{\varepsilon}{k}$
Energy	T	$\mu / \text{Pr} + \mu_t / \sigma_T$	S_T
Smoke	Y	$\mu / \text{Sc} + \mu_t / \sigma_Y$	S_Y

where

$$P = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j}$$

$$G_B = -\frac{\mu_t}{\sigma_T} \frac{1}{\rho} g_i \frac{\partial \rho}{\partial x_i}$$

$$\mu_t = c_\mu \rho \frac{k^2}{\varepsilon}$$

$$c_1 = 1.44, \quad c_2 = 1.92, \quad c_\mu = 0.09, \quad \sigma_k = 1.0, \quad \sigma_\varepsilon = 1.3, \quad \sigma_T = \sigma_Y = 0.7$$

Pr = Prandtl number

Sc = Schmidt number

5.1.2 Turbulence Model

The tunnel model is based on the buoyancy-augmented k - ε model. In the k - ε model [3], turbulence is represented by the turbulence kinetic energy, k , and its rate of dissipation, ε . Differential equations of the form of equation 5-1 govern both these turbulence parameters. The source terms for k and ε are given by:

$$S_k = P + G_B - \rho \varepsilon \quad (5-2)$$

$$S_\varepsilon = c_1(P + G_B) \frac{\varepsilon}{k} - c_2 \rho \frac{\varepsilon^2}{k} \quad (5-3)$$

In the above equations, P is the turbulence production due to shear and G_B is the production due to buoyancy, given by

$$G_B = -\frac{\mu_t}{\sigma_T} \frac{1}{\rho} g_i \frac{\partial \rho}{\partial x_i} \quad (5-4)$$

where μ_t is the turbulent viscosity, σ_T is the turbulent Prandtl number for temperature, and g_i denotes the component of the gravitation acceleration along the coordinate direction i . The term G_B describes the exchange of turbulence kinetic energy with the potential energy of the system; it takes on opposite signs in the regions of stable and unstable stratification. This term is negative, acting to suppress turbulence, in regions where the stratified flow is stable. In regions of unstable stratification, this term is positive and leads to augmentation of turbulence.

The turbulent viscosity is given in terms of k and ε as

$$\mu_t = c_\mu \rho \frac{k^2}{\varepsilon} \quad (5-5)$$

and the turbulent diffusion coefficient for a scalar (e.g., temperature, species concentration) is calculated as

$$\Gamma = \frac{\mu_t}{\sigma_t} \quad (5-6)$$

where σ_t is the turbulent Prandtl or Schmidt number.

5.1.2 (a) Treatment of the Wall Boundary

In turbulent flow, large gradients are encountered in the vicinity of walls. In addition, the flow conditions in these regions are dominated by molecular diffusion. Accurate representation of the wall boundary layers would require a very large number of grid points near the walls and the resulting computational effort will be excessive. In engineering models, such a detailed calculation is avoided by assuming a Couette flow approximation close to the wall. In this approach, the first grid point next to the wall is placed in the fully turbulent region, and the universal logarithmic laws of wall (“wall functions”) are used to describe the momentum and convective heat/scalar fluxes between the near-wall grid point and the wall surface.

Momentum Equations

For the momentum equations, the wall shear stress is given by

$$\tau_w = \frac{\mu \cdot u_p}{y_p} \left(\frac{y^+}{u^+} \right) \quad (5-7)$$

where μ is the molecular viscosity, u_p is the velocity component parallel to the wall at the near-wall grid point located at a distance y_p from the wall. u^+ and y^+ are the dimensionless velocity and distance and are defined as

$$u^+ = \frac{1}{\kappa} \ln(E \cdot y^+) \quad (5-8)$$

$$y^+ = \frac{\rho \cdot k^{1/2} \cdot c_\mu^{1/4} \cdot y_p}{\mu} \quad (5-9)$$

where κ is the von Karman constant and the constant E accounts for the effects of the laminar sub-layer. The relationship given by equation 5-8 is used if $y^+ > 11.5$; at the near-wall grid points where $y^+ < 11.5$, u^+ is set equal to y^+ . The value of E depends on the wall roughness and is obtained as follows [4]:

$$E = 9 \quad \text{for } k^+ < 9.4 \quad (5-10a)$$

$$E = \frac{210}{(k^+)^{1.409}} \quad \text{for } 9.4 < k^+ < 134 \quad (5-10b)$$

$$E = \frac{28.5}{k^+} \quad \text{for } k^+ > 134 \quad (5-10c)$$

In equation 5-10, k^+ is the non-dimensional surface roughness height and is defined as

$$k^+ = \frac{\rho \cdot k^{1/2} \cdot c_\mu^{1/4} \cdot e}{\mu} \quad (5-11)$$

where e is the roughness height.

General ϕ Equation

For dependent variables such as temperature or species concentration, the wall flux is given by

$$J_w = \left(\frac{\mu \cdot y^+}{\Theta} \right) \frac{(\phi_w - \phi_p)}{y} \quad (5-12)$$

where the subscripts W and P refer to the wall and the first grid point away from the wall, located at a distance of y from the wall, and Θ is the dimensionless definition of ϕ defined as:

$$\Theta = \sigma_t \left[\frac{1}{\kappa} \ln(E \cdot y^+) + P_j \right] \quad (5-13)$$

In the above equation, P_j represents the resistance of the laminar sub-layer to the transport of ϕ and is dependent on the molecular and turbulent Prandtl/Schmidt numbers as well as the surface roughness.

P_j is calculated using the following expression [4]:

$$P_j = 3.15 \cdot \sigma^{0.695} \left(\frac{1}{E} - \frac{1}{E_M} \right)^{0.359} + \left(\frac{E}{E_M} \right)^{0.6} P_M \quad (5-14)$$

In equation 5-14, the subscript M refers to the values for a smooth surface. The values of E are calculated using equation 5-10. The expression for P_M (P_j for a smooth surface) is as follows

$$P_M = 9 \left(\frac{\sigma}{\sigma_t} - 1 \right) \left(\frac{\sigma}{\sigma_t} \right)^{-1/4} \quad (5-15)$$

where σ and σ_t are the molecular and turbulent Prandtl numbers, respectively.

Turbulent Parameters

The diffusion flux of the turbulence kinetic energy is set equal to zero at the wall. The near-wall value of rate of dissipation is deduced from the assumption that the length scale in this region is linearly proportional to the distance from the wall.

5.1.3 Treatment of Conjugate Heat Transfer

A conjugate heat transfer problem involves consideration of both conduction in the solid and convection in the fluid, with a proper matching at the fluid-solid interface. In the tunnel ventilation model, the viscosity (diffusion coefficient for the momentum equations) within the solid regions is specified as a large value and the velocity components are set equal to zero. For solving the temperature equation, the true conductivities of the solid materials are specified in the solid regions of the computational domain; for turbulent flows, the conductivity in the fluid region is obtained from the turbulence model. The problem is solved as a convection-diffusion problem throughout the entire domain, but since the velocities in the solid regions are zero, a pure conduction calculation is performed in these regions. The resulting solution gives temperature distributions in the solid and in the fluid, with proper matching at the interface. Within this framework, isothermal blockages can easily be represented as regions of very high thermal conductivity and insulated blockages as regions of very low (zero) conductivity.

5.1.4 Boundary Conditions

5.1.4 (a) 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 is assumed to be zero and no other information is needed. The normal velocity components at an outflow boundary are adjusted to ensure the overall conservation of mass within the domain. For situations in which the value of pressure is specified at the portals, the conditions within the tunnel determine whether the fluid enters or exits through portions of a portal. The given value of pressure is interpreted as total pressure at the inflow points and static pressure at the outlet points. At the inflow points on a free boundary, the ambient values of temperature and other scalars (e.g., the concentration of the combustion products, turbulence parameters) are specified; at the outflow points, the diffusion flux is set equal to zero.

In the computational model, a provision has also been provided for the introduction of entrance loss. The presence of wind can be modeled by specifying a differential pressure between the portals.

5.1.4 (b) Tunnel Walls

At a solid-fluid interface, the wall-function approach discussed in Section 5.1.2 is employed. For the momentum equations, the no-slip condition is imposed on the solid surfaces. For the species concentrations, the diffusion flux into the wall is zero. For temperature, additional considerations apply.

For the calculation of the temperature field, the tunnel walls can be specified as isothermal surfaces or as surfaces with prescribed values of heat flux. In principle, the boundary values of temperature and heat flux can vary with space and time.

The tunnel walls and internal blockages can be specified as hydraulically smooth or rough surfaces. The wall functions are suitably modified to account for the effect of wall roughness on momentum and heat transfer.

5.2 Representation of Fire

In the tunnel ventilation model, fire is represented as a source of heat and mass. The model does not simulate the combustion process. Instead, the heat release rate due to combustion is prescribed as a volumetric heat source in a *postulated* fire region. Thus 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 combustion efficiency (η), as

$$Q = \dot{m}_{fu} H_{fu} \eta \quad (5-16)$$

In the fire region, the additional volumetric source term in the mass continuity equation is given by

$$S_{m,fire} = \frac{\dot{m}_{fu}}{V_{fire}} \quad (5-17)$$

where V_{fire} is the volume of the fire region. For the energy equation, the additional source term is taken as

$$S_{h,fire} = \frac{Q_C}{V_{fire}} \quad (5-18)$$

where Q_C is the energy transported away by convection; the expression for Q_C depends on the model for radiation heat transfer and is discussed in the next section.

5.3 Representation of Radiation Heat Transfer

The tunnel model includes two options for the representation of radiation transport in the tunnel. In the *radiative fraction* approach, thermal radiation in the participating medium is ignored and a fixed fraction of the total heat released in the fire is assumed to be lost to the surroundings without affecting the temperature distribution within the tunnel; the remaining energy is transported away by the fluid. Experiments on diffusion flames indicate that the radiation fraction typically lies in the range 0.2 to 0.4 [5]. With this approach for radiation heat transfer, the energy convected by the fluid is given by

$$Q_C = \dot{m}_{fu} H_{fu} \eta (1 - \chi_R) \quad (5-19)$$

where χ_R is the radiative fraction.

In the *detailed radiation treatment* approach, the six-flux radiation model [6] is used for detailed calculation of thermal radiation in the participating medium. With this approach, the entire heat released in the fire region is introduced as a source in the energy equation, and the radiation model determines the energy lost to the tunnel walls. The detailed model however needs additional information about the radiation properties of the medium and the tunnel walls. The details of the six-flux radiation model are presented in the next section.

5.3.1 Six-Flux Model for Radiation Heat Transfer

5.3.1 (a) Governing Equations

In the six-flux model for radiation heat transfer, the radiation field is assumed to be represented by six fluxes at each point in space. For a Cartesian coordinate system, these are:

$$I_x^+, I_x^-$$

$$I_y^+, I_y^-$$

$$I_z^+, I_z^-$$

In each of these directions, each flux is assumed to change according to local emission and absorption + scattering. If I_i^+ is the flux in the positive i direction and I_i^- is the flux in the negative i direction, then the governing equations are

$$\frac{dI_i^+}{dx_i} = -(a + s)I_i^+ + aE_b + \frac{s}{6} \sum_{k=1}^3 (I_k^+ + I_k^-) \quad (5-20)$$

$$\frac{dI_i^-}{dx_i} = (a + s)I_i^- - aE_b - \frac{s}{6} \sum_{k=1}^3 (I_k^+ + I_k^-) \quad (5-21)$$

where a and s are local absorption and scattering coefficients and E_b is the black body emissive power. These equations can be combined to give

$$\begin{aligned} \frac{d}{dx_i} \left(\Gamma_R \frac{d}{dx_i} (I_i^+ + I_i^-) \right) &= -S_i \\ &= (a + s)(I_i^+ + I_i^-) - 2aE_b - \frac{s}{3} \sum_{k=1}^3 (I_k^+ + I_k^-) \end{aligned} \quad (5-22)$$

where

$$\Gamma_R = \frac{1}{a + s} \quad (5-23)$$

Defining a composite variable

$$R_i = I_i^+ + I_i^- \quad (5-24)$$

Equation 5-22 can be written as

$$\frac{d}{dx_i} \left(\Gamma_R \frac{dR_i}{dx_i} \right) + S_i = 0 \quad (5-25)$$

Equation 5-25 has the same form as the generic conservation equation (5-1) solved in the COMPACT programs.

The radiation contribution to the enthalpy source term for each control volume is calculated as

$$S_{h,rad} = - \sum_{i=1}^3 S_i = a \sum_{i=1}^3 (I_i^+ + I_i^-) - 6aE_b \quad (5-26)$$

5.3.1 (b) Treatment of Wall Boundaries

At the walls, the radiation heat flux lost from the fluid will depend on the wall temperature, wall emissivity, and reflectivity. The radiation intensity leaving the wall has contributions from the wall emission and reflection. Thus, the intensity leaving a wall surface located at $X = 0$ can be expressed as

$$I_x^+ = \varepsilon_w \cdot \sigma \cdot T_w^4 + \rho_w \cdot I_x^- \quad (5-27)$$

where ε_w and ρ_w are wall emissivity and reflectivity. Since the transmissivity of the surfaces is zero, it follows that

$$\rho_w = 1 - \varepsilon_w \quad (5-28)$$

The net heat flux into the wall is given by

$$q_w = I_x^- - I_x^+ \quad (5-29)$$

In terms of the composite variable R_x , equation 5-29 can be written as

$$q_x = - \frac{\varepsilon_w}{2 - \varepsilon_w} (2\sigma \cdot T_w^4 - R_x) \quad (5-30)$$

Equation 5-30 gives the amount of radiation absorbed by the walls.

In the present implementation of the six-flux model, the energy absorbed by the walls is assumed to be lost from the tunnel walls without affecting the temperature distribution in the walls. This assumption is reasonable because the period of time following the start of fire in the tunnel is relatively short.

5.4 Representation of Smoke

In the tunnel ventilation model, a separate conservation equation is solved for smoke. The conservation equation for smoke 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 and is based on the assumption of complete combustion. It is given by

$$\dot{m}_{smoke} = \dot{m}_{fu} (1 + s) f_{smoke} \quad (5-31)$$

where s is the stoichiometric ratio (kg of air / kg of fuel) for the fuel. In equation 5-31, a factor, f_{smoke} , has been introduced. The value of this factor depends on the interpretation of smoke: whether it is considered as the products of combustion or as the “particulate” smoke. For the former definition, f_{smoke} is unity; for the latter it can be interpreted as the smoke potential of the fuel. The volumetric source term in the smoke equation is given by

$$S_{m,smoke} = \frac{\dot{m}_{smoke}}{V_{fire}} \quad (5-32)$$

It should be noted that due to the differences in the boundary conditions at the tunnel walls and the nature of the source terms, the distributions of temperature and the products of combustion, in general, will not be identical.

The model for smoke movement outlined here does not account for soot formation. A comprehensive soot model would involve a reaction model for the formation of soot and also a model for detailed gas-phase chemistry. Such a detailed model for soot was considered beyond the scope of this project.

5.5 Representation of Jet Fan

A jet fan is represented as a constant volumetric flow rate device. Here it should be recognized that the constant flow rate is applied only to the flow actually going through the jet fan. The flow *induced* by the jet fan in the entire tunnel is not held constant. Indeed, the interaction of the jet (produced by the jet fan) with the surrounding air decides how much flow gets induced in the tunnel. The induced flow also depends on the size, shape and length of the tunnel and on the presence of any obstacles.

In the model, a jet fan is simulated by a combination of sources and sinks. The specific details are as follows:

- The control volumes within the fan region are blocked-off, that is, they are treated as solid. The diffusion coefficients for temperature and species concentrations in the fan region are set equal to zero so that there is no interaction between the fluid within the fan and the fluid on the outside.

- Mass and momentum sources (and sources for turbulence parameters and energy) are introduced in the fluid control volumes adjacent to the discharge end of the fan. The strengths of the mass and momentum sources are determined from the fluid density on the intake end of the jet fan, the fan capacity and the discharge velocity. The sources for k and ε are calculated using the following incoming values for the jet leaving the fan

$$k = I \cdot u_{jet}^2 \quad (5-33a)$$

$$\varepsilon = \frac{k^{1.5}}{D} \quad (5-33b)$$

where u_{jet} is the discharge velocity, I is related to the turbulence intensity and D is the jet fan diameter.

For the energy equation, the sources at the discharge end are calculated using the fact that there is no energy addition to the fluid within the fan. Thus, the total enthalpy entering the fan from the intake end is introduced as source at the discharge end. A similar practice is used for smoke.

- Mass sinks are introduced in the fluid control volumes next to the intake end of the fan.

5.6 Representation of Ventilation Ducts

In transverse ventilation systems, air is supplied to and exhausted from the tunnel through ventilation ducts. The supply and exhaust rates are not known *a priori*; these depend not only on the characteristics of the duct systems but also on the conditions within the tunnel. The model for such systems, therefore, requires a methodology for the calculation of the supply and exhaust rates and the interaction between the ventilation ducts and the tunnel.

In the model, the ventilation duct system is represented as a network of links and nodes. The values of pressure, temperature, and concentrations 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 provided 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. A brief description of the fundamentals of the network model is presented next. This is followed by the specific details related to the tunnel ventilation model.

5.6.1 Basic Details of the Network Model

5.6.1 (a) Governing Equations

The basic governing equations in a 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

$$\sum_{j=1}^J (\rho_{ij} Q_{ij}) = -m_i \quad (5-34)$$

J is the total number of links associated with node i . Further, ρ is the density, Q is the volumetric flow rate, and m is the external mass flow into node i . In equation 5-34 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 can be expressed as

$$\Delta p_{ij} = L(Q_{ij}) + F(Q_{ij}) \quad (5-35)$$

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

$$\sum_{j=1}^J (\rho_{ij} Q_{ij} H_{ij}) = S_i^H \quad (5-36)$$

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 .

5.6.1 (b) 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

$$L(Q_{ij}) = R |Q_{ij}| Q_{ij} \quad (5-37)$$

The heat loss from a link is calculated as

$$q_{ij} = h_{ij} A_{ij} (T_f - T_w) \quad (5-38)$$

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 [7]).

5.6.1 (c) Fan Pressure Rise

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

$$\Delta p_f = a + bQ + cQ^2 + dQ^3 \quad (5-39)$$

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

In the implementation of the network model, fluid and thermal inertia terms are not included in the governing equations. 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 given relationship. The duration of the fan run-up/down is provided as input and is used to compute a factor, f , as follows:

$$f = 1 - e^{-K \left(\frac{t-t_e}{t_r} \right)^2} \quad \text{for, } t_e \leq t \leq t_e + t_r \quad (5-40)$$

where K is equal to $\ln(100)$, t is the simulation time within the run-up/down period, t_e is the time of the event (i.e., fan operational change), and t_r is the duration of the run-up/down.

This factor is applied to the difference in fan pressures at the start and end of the run-up/down period, as follows:

$$\Delta p_{f,t} = \Delta p_{f,1} + (\Delta p_{f,2} - \Delta p_{f,1})f \quad (5-41)$$

where $\Delta p_{f,t}$ is the fan pressure during the run-up/down period, and $\Delta p_{f,1}$ and $\Delta p_{f,2}$ are the fan pressures at the start and end of the run-up/down period, respectively.

5.6.1 (d) 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.

5.6.1 (e) Solution Procedure

The governing equations are discretized using the well-established procedure described by Patankar [2]. 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.

5.6.2 Network Model for Transverse Ventilation Systems

As stated earlier, the modeling of transverse ventilation systems requires the calculation of the supply and exhaust rates, which depend on the conditions within the tunnel as well as on the characteristics of the duct system. In the model, a field model (based on COMPACT) is used to simulate the conditions in the tunnel, and a network model is used for the ventilation ducts. These two 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 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.

5.6.2 (a) Representation of Tunnel Conditions and Tunnel-to-Duct Communication

To model the interaction between the tunnel and ventilation ducts properly, a mechanism for communication between the duct system and the tunnel was devised. This mechanism provides a methodology for the network representation of slots on the tunnel walls (through which the ducts communicate with the tunnel) and for the representation of the tunnel conditions. It should be noted that the dimensions of the slots might be much smaller compared to the near-boundary grid size 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 employed in the model is illustrated via the sample network for an exhaust duct system shown in Figure 5.6-1. For each ventilation duct, the tunnel under the duct is divided into a number of zones (*tunnel zones*) in the longitudinal and the transverse or vertical (depending on the location of the duct) directions. A tunnel zone may include a number of control volumes of the tunnel (field) 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 zone are represented by average values of pressure and temperature in the zone. These values are calculated from the distributions given by the field model for the tunnel. 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 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 additional tunnel nodes and the tunnel-to-duct links in the network model allow interaction between the tunnel and the duct.

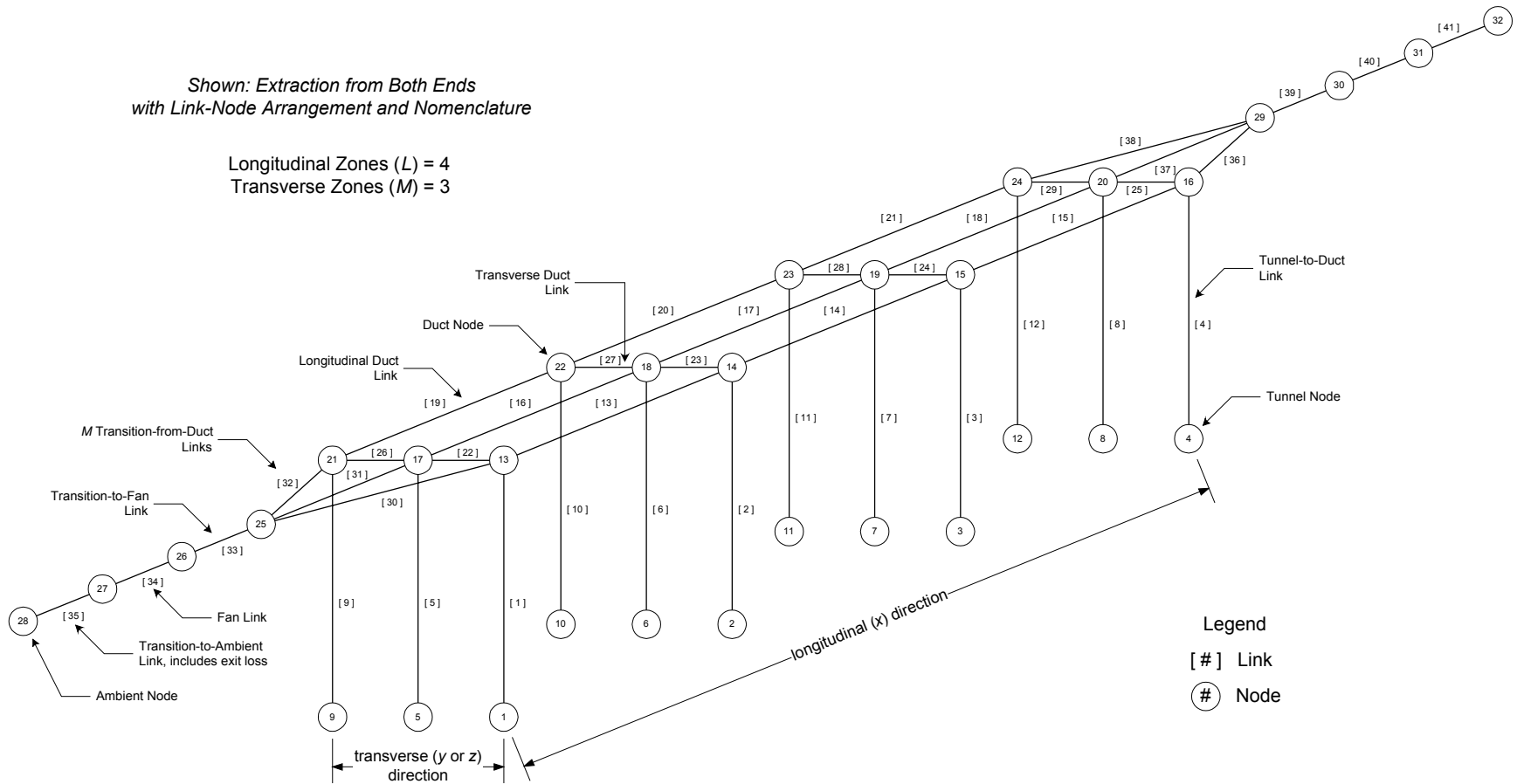


Figure 5.6-1 Example of an Exhaust Duct Network

5.6.2 (b) Overall Solution Procedure

For a given set of conditions in the tunnel and other ambient conditions, the network model gives the flow rates for various links, including the special tunnel-to-duct links, in the network. The mass flow rates through the tunnel-to-duct links are introduced as mass sources or sinks in the field model for the tunnel. The tunnel model, in turn, provides the boundary conditions for the network model. The coupling between the two 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.
7. Repeat steps 4 through 6 until convergence is achieved.

As previously stated, fluid and thermal inertia terms are not included in the governing equations in the implementation of the network model. 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. The fan run-up/down feature tempers the time-related effect of not including fluid inertia in the network model.

5.7 Numerical Solution Method for the Field Model

In this section, a brief overview of the numerical method used for the solution of the governing equations is presented. Complete details of the method and its implementation are available in the Reference Manual for COMPACT-3D [1].

5.7.1 Domain Discretization and Location of the Dependent Variables

The tunnel ventilation model is based on COMPACT-3D, a general-purpose computer program for the analysis of fluid flow, heat transfer, and related processes. COMPACT-3D is based on the finite-volume representation of the physical laws governing the conservation of mass, momentum, energy, and other quantities. The region of interest is divided into a number of non-overlapping control volumes and a grid point is assigned at the center of each control volume. In the COMPACT programs, a staggered grid arrangement is used for the location of the dependent variables. The scalar quantities (e.g., pressure, temperature, and species concentrations) are stored at the grid points, which are located at the centers of the control volumes, and the velocity components are located at the control-volume faces. This arrangement maintains strong coupling between the velocity and pressure fields and prevents the appearance of a checker board pressure field.

5.7.2 Derivation of the Discretization Equations

The governing differential equations are integrated over their control volumes to give a set of algebraic equations involving the unknown values of the dependent variables. The resulting numerical solution satisfies the conservation principles over each control volume and, hence, over the entire computational domain. A novel feature of the COMPACT programs is the use of weighted harmonic mean averaging to calculate the diffusion coefficient at a control-volume face from the values specified at the main grid points. The resulting formulation can handle large spatial disparities in the diffusion coefficients and also simplifies the treatment of conjugate heat transfer problems.

A typical control volume is shown in Figure 5.7-1. The upper case letters denote the grid points and the lower case letters refer to the faces of the control volume. The discretization procedure requires an assumption regarding the variation of the dependent variable between the grid points. In the COMPACT programs, the power-law differencing scheme is used. The coupling between the velocity and pressure fields is handled using the SIMPLER algorithm. For a time-dependent calculation, the COMPACT programs use the implicit scheme for the discretization of the unsteady terms in the equations. With these practices, the final discretization equation for the unknown ϕ_p is given by

$$a_p \phi_p = \sum_{nb} (a_{nb} \phi_{nb}) + b \quad (5-42)$$

where a 's represent the discretization coefficients, the subscript nb refers to the neighboring nodes and b represent the influences of sources and sinks for the control volumes associated with grid point P .

Nonlinear source terms are expressed in a linearized form as

$$S = S_C + S_P \phi_p \quad (5-43)$$

S_p must not be positive to avoid computational instabilities and the possibility of unrealistic solutions.

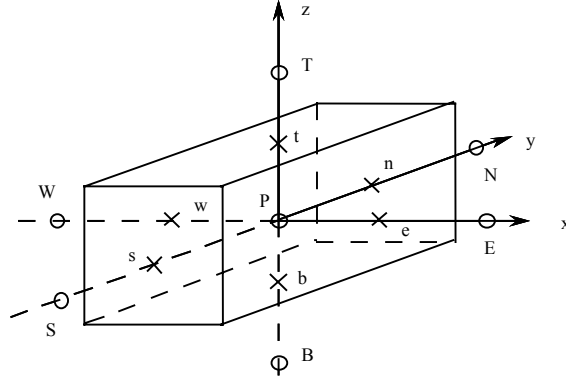


Figure 5.7-1 Control Volume Around the Grid Point P

The coefficients a_{nb} represent the combined influence of diffusion and convection through the control-volume face and depend on the choice of the differencing scheme. For the power-law scheme, for example,

$$a_E = D_e A^+ (|P_e|) + \max[0, -F_e] \quad (5-44)$$

$$a_W = D_w A^+ (|P_w|) + \max[0, F_w] \quad (5-45)$$

where the function $\max[a, b]$ denotes the greater of a and b , D is the conductance, F is the mass flow rate and P is the Peclet number at the control-volume face. These quantities are defined as

$$D = \Gamma \frac{A}{\delta x} \quad (5-46)$$

$$F = \rho \cdot u \cdot A \quad (5-47)$$

$$P = \frac{F}{D} \quad (5-48)$$

The function $A^+ (|P|)$ is given by

$$A^+ (|P|) = \max[0, (1 - 0.1|P|)^5] \quad (5-49)$$

The source term b in equation 5-42 is given by

$$b = S_C \cdot \Delta V + a_p^o \cdot \phi_p^o \quad (5-50)$$

where the superscript o refers to the “old” known values at time t while all other variables are unknown at the new time $t + \Delta t$

$$a_p^o = \rho_p^o \left(\frac{\Delta V}{\Delta t} \right) \quad (5-51)$$

and

$$a_p = \sum_{nb} (a_{nb}) + a_p^o - S_p(\Delta V) \quad (5-52)$$

5.7.2 (a) Treatment of Mass Sources and Sinks

If there are mass sources (sinks) in the domain, their influence (except in the continuity equation) is introduced through *additional* sources as follows:

$$S_C = \max[0, \dot{m}'''] \cdot \phi_{in} \quad (5-53)$$

$$S_p = -\max[0, \dot{m}'''] \quad (5-54)$$

where \dot{m}''' is the volumetric mass source and ϕ_{in} is the value of ϕ in the source fluid.

5.7.2 (b) Solution of the Discretization Equations

The discretization equations are solved using the line-by-line TriDiagonal-Matrix Algorithm (TDMA), supplemented by a block-correction procedure; for complete details see the Reference Manual for COMPACT-3D [1], Patankar [2], and Patankar et al. [8]. In the line-by-line method, the TDMA is used to solve for ϕ 's along a grid line (e.g., in the y -direction), by assuming that the ϕ 's along the neighboring lines (i.e., in the x - and z -directions) are known from their latest values. This procedure is repeated for all the lines in one direction, and the procedure is repeated for the lines in the other two directions.

5.7.3 Summary of the Overall Calculation Procedure

The overall calculation procedure for the solution of the field equations involves the following main calculation steps. Complete details and corresponding equations are available in the Reference Manual for COMPACT-3D.

- (i) Guess the values of all the dependent variables.
- (ii) Calculate the coefficients in the momentum equations and solve for the velocity components.
- (iii) Evaluate the mass source for the pressure-correction equation and solve for pressure corrections.

- (iv) Using the pressure-correction values obtained in Step (iii), correct the existing pressure and velocity fields.
- (v) Using the currently available pressure values, solve the momentum equations to get new velocity fields.
- (vi) Using the velocity components obtained in Step (v), evaluate the mass source term in the continuity equation and solve for the second set of pressure corrections.
- (vii) Using this pressure-correction field, correct the velocities obtained in Step (vi).
- (viii) If the values of other ϕ 's such as temperature, concentration, turbulence parameters, etc. influence the solution of the velocity field, solve the conservation equations for these ϕ 's.
- (ix) Return to Step (ii) with the new values of all the ϕ 's including the velocity components and pressure and repeat the procedure until convergence.
- (x) If there are dependent variables ϕ that require the velocity field for their solution but do not influence the velocity field, solve the discretization equations for these ϕ 's.

For steady-state problems, the above sequence of steps is iterated until convergence is achieved. For linear unsteady problems, this sequence can be considered as the solution for one time step. For nonlinear unsteady problems, the general iteration loop described above is repeated *within each time step* until convergence. The converged results at the end of a time step represent the *old* time values and also the guess values for the next time step. The time is then advanced by an appropriate time interval Δt . This process is repeated for a pre-specified number of time steps.

5.7.4 Nonlinearities, Interlinkages and Underrelaxation

5.7.4 (a) Treatment of Nonlinearities and Interlinkages

In COMPACT-3D, various interlinkages and nonlinearities in the governing equations are handled via iterations. At any given stage, the discretization coefficients are calculated from the current estimates of all the ϕ values. The solution of the discretization equations gives an improved estimate which is used to recalculate the coefficients. When the change in all of the ϕ 's from iteration to iteration is sufficiently small, the final solution is reached.

5.7.4 (b) Use of Underrelaxation

It does not always follow that successive iterations will lead to a converged solution. At times, the values of ϕ oscillate or drift continuously. Such divergence of the iterative process must be avoided. Although for linear equations, the algorithm used in COMPACT-3D is guaranteed to converge, this is not true for nonlinear and coupled problems. Although the solution algorithm has been designed to minimize the chances of divergence, additional techniques are often needed to improve convergence. One such

strategy is to slow down the changes in the coefficients from iteration to iteration by controlling the changes in the ϕ values. This is known as underrelaxation.

Underrelaxation can be introduced into the discretization equations in a number of ways. The practice used in COMPACT-3D is to write the general discretization equation

$$a_p \phi_p = \sum_{nb} (a_{nb} \phi_{nb}) + b \quad (5-55)$$

as

$$(a_p + i) \phi_p = \sum_{nb} (a_{nb} \phi_{nb}) + b + i \phi_p^* \quad (5-56)$$

where i is the so-called inertia. Further, the inertia is chosen as

$$i = \left(\frac{1-\alpha}{\alpha} \right) \sum_{nb} (a_{nb}) \quad (5-57)$$

where α is the relaxation factor. When $\alpha = 1$, no underrelaxation is introduced. As α approaches zero, the changes in ϕ are slowed down. If $\alpha > 1$, the effect is overrelaxation, in other words, the changes in ϕ are exaggerated. In general, $\alpha > 1$ is not recommended.

In addition to the dependent variables, other quantities are also underrelaxed. The formula for underrelaxing an auxiliary variable such as the density ρ is

$$\rho = \alpha \cdot \rho_{new} + (1-\alpha) \rho_{old} \quad (5-58)$$

This type of underrelaxation weakens the links between different dependent variables. For example, if the turbulent viscosity is underrelaxed, the changes in the turbulence parameters cannot strongly influence the velocity components. If the buoyancy source term is underrelaxed, the changes in the temperature field do not significantly affect the velocity field.

Although different forms of underrelaxation promote convergence, there is no unconditional guarantee that the iterative solution procedure will always converge for all kinds of nonlinearities and interlinkages. Also, there are no general formulas for choosing the optimum values of the underrelaxation factors. A trial-and-error procedure is generally required to devise an underrelaxation procedure that leads to a converged solution.

5.7.5 Convergence Criterion

Since the solution procedure used in COMPACT-3D is iterative, some criterion must be used to decide when a converged solution has been reached. A suitable convergence criterion depends on the nature of

the problem under consideration. A common practice to determine convergence is to monitor the changes in the grid-point values of the dependent variables from iteration to iteration. However, this practice may lead to erroneous conclusion when heavy underrelaxation is used. The underrelaxation forces the changes to be small; this may cause an illusion of convergence when the computed solution is far from being converged. A more appropriate method of monitoring convergence is to examine how well the discretization equations are satisfied by the existing values of the dependent variables. For each grid point, a residual R can be computed as

$$R = \sum_{nb} (a_{nb}\phi_{nb}) + b - a_p\phi_p \quad (5-59)$$

For a perfectly converged solution, R will be zero everywhere. In the model, the convergence criterion is based on the normalized sum of $|R|$, defined as

$$R_{norm} = \frac{|R|}{\max[a_p\phi_p, b, a_{nb}\phi_{nb}]} \quad (5-60)$$

where the summation is over all grid points in the domain.

For many transverse ventilation situations, the flows are buoyancy-dominated. When the buoyancy forces become strong, the flow has a tendency to become unsteady and may not reach a steady-state. When such flows are solved on a steady-state basis, it may not be possible to achieve a fully converged solution. That is, the residuals of the various equations stop decreasing beyond certain values and show a tendency for mild oscillations. Likewise, minor oscillations may be also present in the local values of the dependent variables. However, the overall features of the flow do not change appreciably from one iteration to the next. In judging convergence for transverse ventilation, therefore, one should not rely solely on the spot values and the residuals, but should also look at the field distributions at different iteration levels.

References Cited

1. Innovative Research, Reference Manual for COMPACT-3D, Innovative Research, Inc., Minneapolis (1998).
2. Patankar, S.V., *Numerical Heat Transfer and Fluid Flow*, Taylor and Francis (1980).
3. Launder, B.E. and Spalding D.B., The Numerical Computation of Turbulent Flows, *Computer Methods in Applied Mechanics and Engineering*, Vol. 3, pp. 269–289 (1974).
4. Jayatilke, C.L.V., 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 (1969).

5. Markstein, G.H., Relationship Between Smoke Point and Radiant Emission from Buoyant Turbulent and Laminar Diffusion Flames, *20th Symposium (International) on Combustion*, pp. 1055–1061, The Combustion Institute, Pittsburgh (1984).
6. Patankar, S.V. and Spalding, D.B., A Computer Model for Three-Dimensional Flow in Furnaces, *14th Symposium (International) on Combustion*, pp. 605–614, The Combustion Institute, Pittsburgh (1972).
7. Burmeister, L.C., *Convective Heat Transfer*, John Wiley and Sons (1983).
8. Patankar, S.V., Karki, K.C., and Kelkar, K.M., Finite Volume Method, in Johnson, R.W., Ed., *The Handbook of Fluid Dynamics*, CRC Press (1998).