Computer Models
For
Fire and Smoke

Model Name:	SOLVENT
Version:	1.0
Classification:	Field Model (Computational Fluid Dynamics model)
Very Short Description:	A Computational Fluid Dynamics model for the simulation of fluid flow, heat transfer and smoke transport in tunnels. Specifically designed to model longitudinal (jet fans) and transverse ventilation systems.
Modeler(s), Organization(s):	Massachusetts Highway Department; Innovative Research, Inc.; Parsons Brinckerhoff Quade & Douglas, Inc.
User's Guide:	SOLVENT User's Manual and SOLVENT Reference Manual, available as PDF documents with the software
Technical References:	Memorial Tunnel Fire Ventilation Test Program Phase IV Report, Massachusetts Highway Department, Jan. 1999
	S. Levy, J. Sandzimier, N. Harvey, E. Rosenbluth, K. Karki, S. Patankar; CFD Model for Transverse Ventilation Systems; 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.
	K. Karki, S. Patankar, E. Rosenbluth, S. Levy; CFD Model for Jet Fan Ventilation Systems; Proceedings of the 10th International Symposium on Aerodynamics and Ventilation of Vehicle Tunnels; Principles, Analysis and Design; 1–3 November 2000; Boston, USA; Organized and sponsored by BHR Group Limited.
	S. Levy, J. Sandzimier, E. Rosenbluth; Emergency Operating Mode Analysis for the Ted Williams Tunnel;

	Proceedings of the Third International Conference on Tunnel Management 2000; 4-6 October 2000; Sydney, Australia; Organized and sponsored by Independent Technical Conferences Ltd.
	S. Levy, J. Sandzimier; Smoke Control For The Ted Williams Tunnel – A Comparative Study Of Extraction Rate; Proceedings of the 10th International Symposium on Aerodynamics and Ventilation of Vehicle Tunnels; Principles, Analysis and Design; 1–3 November 2000; Boston, USA; Organized and sponsored by BHR Group Limited.
Validation References:	Memorial Tunnel Fire Ventilation Test Program Phase IV Report, Massachusetts Highway Department, Jan. 1999
	S. Levy, J. Sandzimier, N. Harvey, E. Rosenbluth, K. Karki, S. Patankar; CFD Model for Transverse Ventilation Systems; 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.
	K. Karki, S. Patankar, E. Rosenbluth, S. Levy; CFD Model for Jet Fan Ventilation Systems; Proceedings of the 10th International Symposium on Aerodynamics and Ventilation of Vehicle Tunnels; Principles, Analysis and Design; 1–3 November 2000; Boston, USA; Organized and sponsored by BHR Group Limited.
Availability:	Parsons Brinckerhoff Quade & Douglas, Inc.; more details from www.tunnelfire.com or info@tunnelfire.com.
Price:	$820
Necessary Hardware:	A Pentium processor-based personal computer with a CD-ROM drive. Microsoft Windows 95 or later or Windows NT 4.0 Service Pack 3 or later. RAM requirements vary with the size and type of problem, but 128 MB is the minimum recommended amount.
Computer Language:	Fortran/C++

Size: Installation requires 25 MB of hard drive space. Space required for simulation output files varies with size and type of problem.

Contact Information: info@tunnelfire.com or visit the SOLVENT web site at www.tunnelfire.com.

Detailed Description:

SOLVENT is a Computational Fluid Dynamics model for the simulation of fluid flow, heat transfer and smoke transport in tunnels. The tunnel ventilation model employs the buoyancy-augmented k-ε model to represent the turbulent transport and includes component models for jet fans, ventilation ducts, fire, radiation heat transfer, and smoke. SOLVENT is based on the standard finite-volume method and uses a staggered grid arrangement. It uses the SIMPLER algorithm to calculate the pressure field.

SOLVENT was developed in Phase IV of the Memorial Tunnel Fire Ventilation Test Program and is applicable to different ventilation modes, including longitudinal ventilation using jet fans, transverse ventilation, and natural ventilation. It is geared towards individuals (analysts, designers, and tunnel operators) concerned with fire/life safety in tunnels. 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.

SOLVENT includes a customized preprocessor for problem specification. In addition to standard input such as tunnel geometry and boundary conditions, the preprocessor allows specification of internal blockages, jet fans, fire, and ventilation ducts (for a transverse ventilation system).

Jet fans are represented as combinations of mass sources and sinks. For transverse ventilation systems, a flow network model is used to predict the flow rates, temperatures, and pressures in the ducts. The ventilation ducts and the tunnel (which uses the CFD model) are coupled via boundary conditions.

The model has been extensively validated using the test data from the Memorial Tunnel Fire Ventilation Program. The model correctly predicts the airflow generated by a jet fan ventilation system. The model correctly predicts the extent and location of the hazardous region as a function of fire size, fan capacity, and fan operation, under both steady state and transient conditions. Overall, the findings indicate that the model adequately addresses the physics associated with the interaction of a fire and jet fan or transverse ventilation systems, under both steady and transient conditions.