Computer Models
For
Fire and Smoke

Model Name:	MEFE (Model for the prediction of smoke flow in buildings)
Version:	1.0
Classification:	CFD
Very Short Description:	The model is able to solve the flow due to a fire in one or two three-dimensional enclosures. The flow is buoyancy driven and the generated turbulence increases the transfer of field quantities. Turbulence is simulated by the use of standard buoyancy-modified version of κ-ε model with the current wall functions. The developed field code is based on the finite volume technique using a staggered grid. It is able to predict transient phenomena. Predictions are established accounting for the time constant of thermocouples, assuming a first order response, and the radiation effect in order to compare with testing results.
Modeler(s), Organization(s):	João Carlos Viegas, National Laboratory of Civil Engineering
User's Guide:	Not available
Technical References:	VIEGAS, João Carlos Godinho - *Segurança contra incêndios em edifícios. Modelação matemática de incêndios e validação experimental (Fire safety in buildings. Mathematical modelling of fire and experimental validation)*. Lisboa: Instituto Superior Técnico, 1999. Ph. D. thesis.
Validation References:	This field code was validated by experiments carried out in a test facility including two full-scale building compartments. The work program considered extensive testing of twelve fire scenarios combining different geometries of the openings for ventilation, different locations of the fire source and different heat release rates.

Data collected during the tests included measurements of temperature of the fluid and on the inner surface of the walls as well as flow velocity through openings and gas composition. The following documents cite experimental comparisons with the model:

VIEGAS, João Carlos Godinho - *Segurança contra incêndios em edifícios. Modelação matemática de incêndios e validação experimental (Fire safety in buildings. Mathematical modelling of fire and experimental validation)*. Lisboa: Instituto Superior Técnico, 1999. Ph. D. thesis.

VIEGAS, J. C.; M. G. CARVALHO and J. SARAIVA – Flow induced by fire in compartments: tests and simulation. EUROTHERM Seminar #56. Delphi (Greece): National Technical University of Athens, 1998.

VIEGAS, J. C.; M. G. CARVALHO and J. SARAIVA – *Study of natural ventilation of enclosed fires: tests and simulations*. INTERFLAM'99, Proceedings of the eighth international conference. Edinburgh: Interscience Communications, 1999.

VIEGAS, João Carlos; Maria da Graça CARVALHO e Jorge SARAIVA – *Incêndios em edifícios. Validação de um modelo de campo para previsão do escoamento (Fire in buildings. Validation of a field model for the prediction of the flow)*. VI Congresso Nacional de Mecânica Aplicada e Computacional. Aveiro: Universidade de Aveiro, Abril de 2000.

PEREZ DOS SANTOS, P. R., F. MARQUES DA SILVA, J. A. Gil SARAIVA, J. C. VIEGAS e M. G. CARVALHO - *Fluxo de renovação de ar em edifícios (Flow for air renovation in buildings)*. Paper presented to *Congresso Chileno de Ingenieria Mecanica*, Concepción (Chile), Outubro de 1998.

Availability: This is a research model. It is not to sell.

Price: Not applicable

Necessary Hardware: Depends on the characteristics of the available FORTRAN compiler and on the dimension of calculation domain.

Computer Language: FORTRAN 77

Size: Depends on the dimension of the calculation domain.

Contact Information: João Carlos Viegas, National Laboratory of Civil Engineering, Avenida do Brasil, 101, P-1700-066 LISBOA, Portugal, jviegas@lnec.pt, Phone: +351.218443542, Fax: +351.218443024

Detailed Description:

GENERAL FEATURES

The model is able to solve the flow due to a fire in one or two three-dimensional enclosures. The flow is buoyancy driven and the generated turbulence increases the transfer of field quantities. Turbulence is simulated by the use of standard buoyancy-modified version of κ-ε model with the current wall functions. The developed field code is based on the finite volume technique using a staggered grid. It is able to predict transient phenomena. Predictions are established accounting for the time constant of thermocouples, assuming a first order response, and the radiation effect in order to compare with testing results.

FLUID DYNAMICS

Transport equations are solved for velocity, specific enthalpy, kinetic energy of turbulence and energy dissipation rate. The independent variables are the three components of a Cartesian co-ordinate system and the time. Since the turbulent fluctuations of density are not negligible close to the fire, the dependent variables are Favre-averaged.

COMBUSTION AND SOOT FORMATION/CONSUMPTION

The propane combustion is described as simple one-step reaction and fast kinetics is assumed. A transport equation is set for the mixture fraction. The influence of the fluctuation of local properties during the combustion process is introduced through the computation of the variance of mixture fraction, using also a transport equation. Therefore, the dependent variables and fluid properties (namely density, specific heat, mass fraction of oxygen, mass fraction of fuel, mass fraction of combustion products and temperature) are averaged considering a probability density function (expressed by a Beta function).
The soot formation and consumption model adopted was proposed by Fairweather and includes a transport equation for soot mass fraction and another for particle number density. Source terms of the soot mass fraction equation represent the processes of nucleation, surface growth and oxidation. The source terms of particle number density equation represent the processes of nucleation and coagulation. This model requires

laminar flamelet prescriptions related to acetylene mole fractions, that is a pyrolysis intermediate, which is considered to produce the soot. It is assumed a single particle size and a spherical shape for soot particles. The local temperature and other species mass fractions are evaluated by hydrodynamic and combustion equations previously referred to.

RADIATION

Discrete Transfer Method is used to compute the radiation transfer. All properties are assumed to be uniform all over a cell. For the calculation of radiative properties of the fluid, a mixed grey gas/soot assumption was made. The absorption coefficient is a function of temperature and fluid composition and its estimation is based on spectral absorption coefficients.

BOUNDARY CONDITIONS

The use of wall functions for the shear stress, enthalpy flux and k-ε boundary conditions avoid the use of a fine mesh near the solid boundaries. At the wall surface the convective heat flux is added to the radiative heat flux and a finite difference transient one-dimensional model solves the wall heat conduction. In the case of the openings, the domain of calculation is extended to the outside and a mass balance is imposed at the boundaries of every compartment. In the interface between the two compartments (if applicable) the information about relevant quantities is exchanged.

METHOD OF SOLUTION

The coupling between pressure and velocity is solved using the SIMPLER algorithm. To solve the flow problem, a Cartesian mesh is used. In order to reduce the dependence of the final solution from the initial mesh it is adopted a strategy of progressive refinement and subsequent mesh adaptation. The mesh is refined along the calculation through the addition of new control volumes in the planes of the domain where the error is maximum. The error is considered proportional to the local second derivative of the pressure field in the direction normal to the refinement. In the new control volumes the field values calculated before the refinement to the same location are kept as initial guesses. More appropriate field values are found along the iterative process of calculation.
The same error estimator is used to adapt the mesh during the resolution of the flow field. Every plane of the mesh is moved in order to equidistribute the error along the co-ordinate axis. However, this procedure is limited by the geometry of the domain (i.e., position of fire source, position of doors and windows in the boundary, etc.).
In order to speed up the calculation it is possible to consider two sub-domains, one for every compartment. It is used a Schwarz decomposition for scalar quantities and a non-overlapping decomposition for the velocity field, due to the fact that a staggered grid is used. One personal computer is used to solve the fluid flow in each compartment. The computers are linked through a network and they exchange files with relevant boundary information in every single cycle.