

FIRE SIMULATION USING THE FLUENT CFD CODE

The simulation of smoke and airflows generated by fires is considerably important. In particular there is growing demand to demonstrate the likely behaviour of fire and smoke in vehicles, tunnels, station complexes and depots. A review of the range of modelling techniques found the Computational Fluid Dynamics (CFD) codes were suitable for this task. Comparison with published data and with independently validated mathematical models suggested the FLUENT code to be appropriate.

The study examines the suitability of the FLUENT CFD package for simulating fire generated air and smoke flows in buildings and rail vehicles.

The following test cases were completed:

- Fire in a corridor closed at one end
- Fire in a 23m railway carriage
- Room fire model

The report concludes that the results obtained agree well with published experimental data, and that the CFD techniques developed using the FLUENT software provide a viable simulation capability.