VIRTUAL TRAINING

Gernot Beer,
Graz University of Technology, Austria.

ABSTRACT

The training of rescue personnel and operators using computer software and hardware is discussed. A virtual fire emergency simulator developed within a European research project is presented. The simulator can be used by fire fighters and by tunnel operators for training missions.

1. INTRODUCTION

Training of emergency service personnel and tunnel operators is an important aspect in ensuring the safety of tunnels. Often such training is hampered by the fact that limited opportunities exist. For training of fire fighters, either tunnels have to be temporarily closed or special underground facilities used. Also there is the additional environmental problem of making fires for missions either by burning cars or by pan fires. Finally such exercises may damage or dirty the tunnel. Virtual training offers a user-friendly and clean alternative. In a virtual training the tunnel, fire and smoke only exists in computer memory. Virtual exercises have the advantage that they:

- may be repeated as often as necessary
- do not require the closure of tunnels
- do not cause pollution

As an example of virtual training we show here the VIRUALFIRES simulator that has been developed within a European project. In the VIRTUALFIRES simulator, the observer is able to visualise the fire and smoke development and the transport of heat and toxic combustion products inside a tunnel and move through the virtual space in the same way as through a real tunnel. The simulator uses and accesses a database, which contains the results of three-dimensional transient combustion (Computational Fluid Dynamics - CFD) simulations for particular tunnel geometries with associated safety installations, particular fire hazard scenarios, etc. The CFD-results can be displayed on a fixed installation in a CAVE virtual environment or on a portable installation using a PC and a head-mounted display (HMD). Two systems are developed: one where the CFD simulation is pre-calculated, stored into a database and then displayed and another where it is carried out in parallel to the visualisation. In the first system the user will be able to move through the data but will not be able to change the characteristics of the simulation, for example the ventilation characteristics in real time.

In the second system the user may change the properties of the simulation while the data are displayed and observe a real time effect of the changes.

The VIRTUALFIRES system can be used for assessing the fire safety of tunnels, for training of rescue personnel and for planning rescue scenarios and will be able to supplement real fire tests. The end users of this system are rescue organisations such as the fire brigades, tunnel operators and government organisations concerned about tunnel safety. The system can be used for making an objective assessment of the fire safety of existing European tunnels. It can also be used for training drivers on how to behave in the case of a fire emergency in a tunnel.

2. DESCRIPTION

The simulator consists of software components and specialized hardware which allows the three-dimensional visualization of results of transient combustion (Computational Fluid Dynamics - CFD) simulations which can be run concurrently with the visualisation. The general layout is shown in Figure 1. The main components are the user interface, the CFD controller, the data manager, the CFD solver (ICE) and the visualization module.

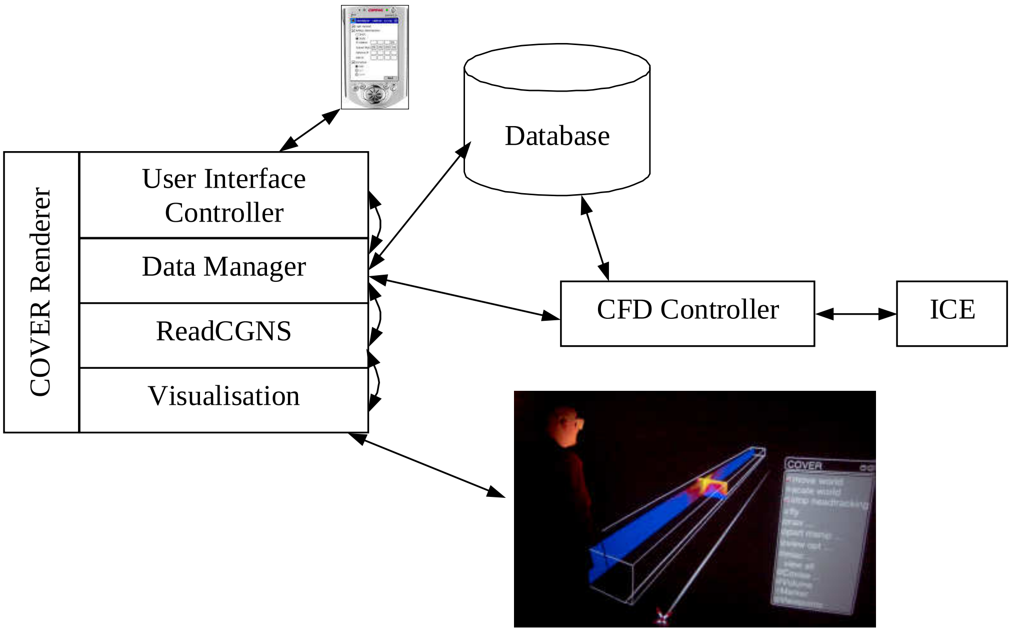

Figure 1. Diagram showing the components of VIRTUALFIRES

2.1 System capabilities

The VIRTUALFIRES system is able to handle tunnels of any cross-section with a variety of installations. The installations can include:

- Fans
- Ventilation inlets and outlets
- Fire extinguishing nozzles
- Escape compartments, exits, lights etc.
- Cars, trucks or rolling stock

The data describing the shape of the tunnel cross-section as well as the fixed installations and the cars etc. are provided in a suitable format (AUTOCAD) and are used to generate the grid for the transient combustion calculation using the program ICE, which will be described later. With the currently available prototype the generation of the grid can, however, not be done completely automatic.

The following additional input data are required:

- Fan capacity in ON position
- Atmospheric pressures at the tunnel portals
- Initial direction of the extinguishing nozzle
- Fire load

Once these data have been provided the user may:

- Define a mission and replay this mission using the forward/stop/rewind/start buttons on the graphic user interface
- During a concurrent session the user may switch on/off existing fans, fire extinguishers and restart a session

The following can be visualised:

- Smoke (using transparency values output by the CFD software) to check visibility
- Isosurfaces of temperature to check survivability
- Streamlines allowing visualisation of the efficiency of the ventilation system

2.2 The user interface

The interface for the control of the interaction with the VIRTUALFIRES system has been realized as a two-dimensional graphical user interface GUI with menus and buttons.

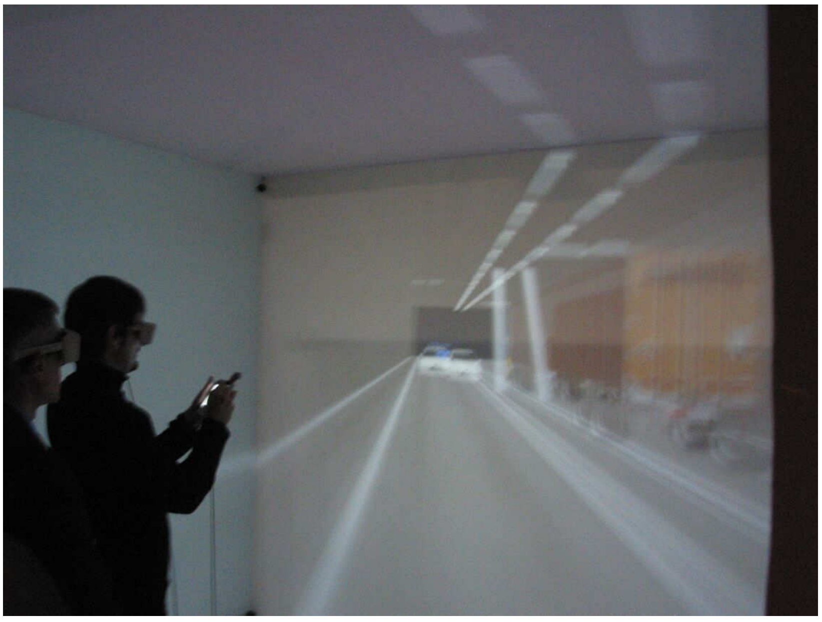

Figure 2. Virtual training in a CAVE environment, interaction with the simulator is realised via a PDA

This GUI can be used in a desktop environment as well as in a portable device (PDA); the graphic appearance is the same in both environments. The PDA used was an HP iPAQ Pocket PC with embedded support for wireless LAN communication. A position tracker is attached to the PDA and therefore this device is used also as a pointer in the 3D space of the CAVE (Figure 2).

The functionality of the GUI allows the user to:

- Load pre-defined configurations and scenarios.
- Describe a mission.
- Start, stop and continue a simulation.
- Adjust the speed of visualization of a simulation, forwards and backwards
- Detach from and attach to an ongoing simulation.
- Stop an ongoing simulation
- Select alternate mission timelines and time steps within them.
- Choose the visualization method.
- Navigate in the 3-D space.

2.3 CFD solver

The Lattice Boltzmann Equation method (LBE) is used and realised in program ICE and its multiprocessor version MPICE. The LBE is a finite difference approximation to the discrete Boltzmann equation (see Luo[1] and Chen and Doolen[2] and Succi[3]. The program MPICE uses a Large Eddy Simulation (LES) model to simulate turbulent flows. The basic idea of a LES is to resolve only the largest unsteady turbulent motions. The role of the small eddies is limited to give the satisfactory role of dissipation that it is concentrated at small scales. The transport equations for scalar quantities, e.g. smoke, are solved by introducing an additional distribution function for each quantity. The particle distribution functions for the scalar quantities evolve on the same lattice than the pressure distribution function for the fluid.

2.3.3 Real time CFD calculation

One of the aims of the project was to achieve real time CFD calculations so that users may immediately see the effect of changes, i.e. of switching on/off fans, of activating fire extinguishers etc.

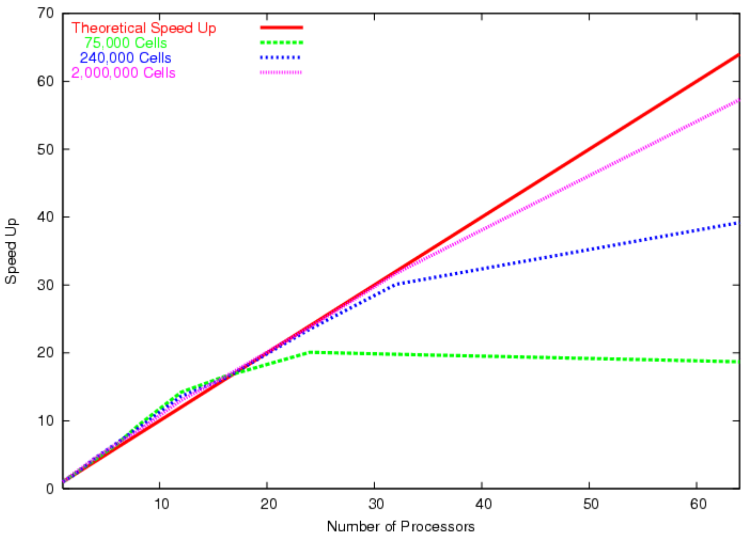

Figure 4. Speed up achieved on LUCIDOR

In the implementation of real time simulation the following has to be considered:

- Real time simulation in connection with real time visualization only makes sense if a suitable number of data sets can be displayed in a very short time. Ideally this would be about 25 data sets per second. In reality one data set which can be transferred to a visualization system and displayed is assumed to be sufficient. In practice only one data set per second can be transferred from the computer to the visualization device via a Gigabit connection.

- From physics and numerics one can conclude that the maximum time step for stable simulations is in the range of 1/50 [s]. Therefore 50 time steps per second real time must be calculated as a minimum. This is only possible using a massive parallel computer and software that has been optimized to this hardware. A supercomputer, the HP Itanium 2 cluster LUCIDOR at the Parallel Dator Centrum at the Kungl Tekniska Högskolan in Stockholm, Sweden[4] was used. LUCIDOR consists of 90 nodes each equipped with two 900 [MHz] Itanium 2 "McKinley" processors. The theoretical peak performance (TPP) of each node is about 7.2 GFLOP/s resulting in a total TPP of 648 GFLOP/s. The processors are interconnected via a full-duplex 2+2 [GBit/s] network connection.

The number of computational cells which can be calculated in real time with this hardware is still small (about 60,000 to 75,000 cells), but in view of the obtained performance data real time simulations consisting of 250,000 cells are feasible in the very near future. However, the number of cells required can be reduced considerably by using 3-D grids only in the area of interest, changing to a 1-D simulation further away.

2.3.4 Coupling of 3-D to 1-D grids

In coupled 3D/1D models, the most interesting part of the flow is simulated in the 3D area i.e. the turbulent flow as well as the flow affected by fans, vents, etc. whereas the area further away is modeled as 1D. The 1D model is defined where the variations of the field values over the cross-section of the tunnel are small. When these variations grow the flow looses its “one-dimensional” nature and cannot be well approximated by such a model. These two requirements, large enough 3D area and “one-dimensionality” of the flow in the 1D area, are satisfied by the following grid expansion/reduction model: At each time step, the variation of the field values is checked at the inner 3D boundaries. A typical scenario for the grid reduction can be the following: In the beginning of the simulation a 3D grid is used in the area of interest. During the analysis the variation of the flow across the section is analyzed. If this variation is below a threshold then the 3D grid is reduced to a 1D grid in the relevant area. The 1D grid is assigned the values (averaged over the cross-section) from the disposed 3D grid. Thus the initial 3D grid is the smallest possible 3D area in the coupled 3D/1D solver. Several tests have been performed where the 3D/1D model was compared to the 3D model and good agreement has been found.

2.4. Visualization methods

The central part of VIRTUALFIRES is the visualization of the results from CFD calculations. The main aim of the visualization system is to give the user the impression of being in a tunnel. An important aspect of the visualization is the change in visibility due to the smoke in the tunnel. Another aspect is the distribution of temperature and toxicity in order to ascertain survivability. In Figures 5 to 7 the different display methods are shown.

Figure 5. Streamlines

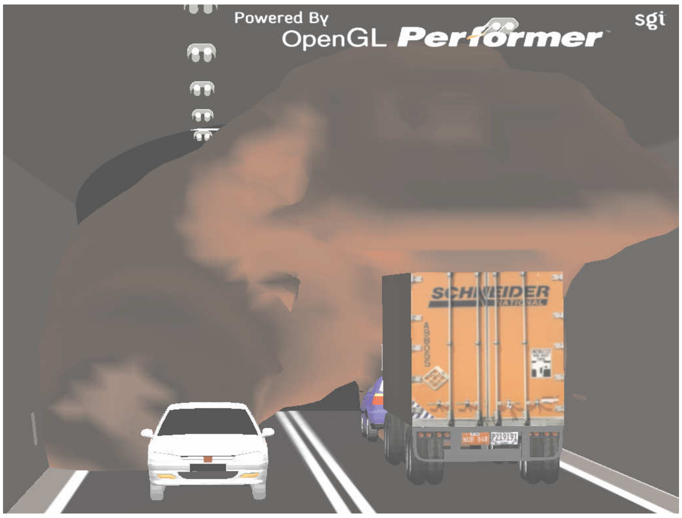

Figure 6. Isosurfaces of temperature

Figure 7. Display of fire and smoke

3. EXAMPLES

For demonstration purposes some well known fire incidents have been calculated with the simulator. These datasets also serve as a base for the verification of the system. The calculated dataset consists of different ventilation scenarios for the Mt. Blanc tunnel in France and the Gleinalm tunnel in Austria. Both tunnels where examined with their former ventilation system and also with the improved ones after the reopening. An example of results is presented in Figures 8. Also a typical subway station in Dortmund has been analysed. In Figure 9 the temperature distribution inside the station during a fire incident on a subway train is shown.

Figure 8. Mt. Blanc Tunnel, spread of smoke(old ventilation system)

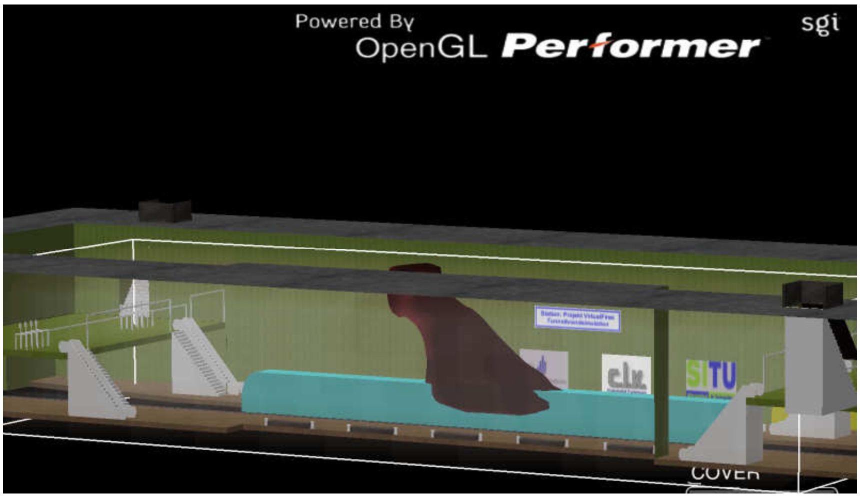

Figure 9. Subway Station Dortmund, temperature distribution,
1 minite after start of fire in train

4. CONCLUSIONS

A simulator was presented which allows fire men to perform virtual training exercises with a head mounted display or in a CAVE environment. The simulator also allows the assessment of the fire safety of existing tunnels and can be used as a tool for designing new tunnels. Currently negotiations are underway with a mayor company, that supplies safety equipment to fire fighters, with a view of commercialising the simulator.

5. ACKNOWLEDGEMENTS

The work reported here was supported by the European Community under the 5th framework (IST) program. The paper contains excerpts from the final report submitted to the European Commission. The coordinator of the project was G. Beer of the Graz University of Technology and the other consortium partners were:

Christian-Doppler-Laboratory for Applied Computational Thermofluiddynamics, Austria
Fraunhofer-Gesellschaft zur Förderung der angewandten Forschung, Germany
Kungl. Tekniska Högskolan, Sweden
Groupement Europèen d`intereêt Economique Alpetunnel and Lyon- Turin Ferroviare, France
Ministere de l'Equipment, des Transport et du Lgement, France
Stadt Dortmund, StA 37 Feuerwehr, Germany
European virtual Engineering , S.A., Spain

6. REFERENCES

1. Luo, L.-S., The Lattice Gas and Lattice Boltzmann Methods: Past, Present, and Future, Proceedings Int. Conf. Appl. Comp. Fluid Dyn., Beijing, China, October 17 – 20, 2000.

2. Chen, S., Doolen, G. D.Lattice Boltzmann Method for Fluid Flows, Annu. Rev. Fluid Mech. 30, 329 – 364 (1998).

3. Succi, S.,The Lattice Boltzmann Equation for Fluid Dynamics and Beyond, Numerical Mathematics and Scientific Computation, Oxford University Press, 2001.

4. http://www.pdc.kth.se.