Fire, smoke and Hazardous Gas Dispersion simulation model based on CFD techniques

The software can be applied to fire propagation, fire suppression and smoke management simulations in any type of buildings as well as for general thermal modeling. It can be used to calculate safety and comfort indicators for the design and construction of buildings. To this end, the software takes into account environmental conditions, HVAC systems, air flow velocity, temperature, relative humidity, thermal radiation and contaminants. All buildings objects like walls, floors etc. can be modelled with their thermal/combustion properties. The same applies to doors, windows and furniture. HVAC devices can transport contaminants and heat through the building and are modelled as specialized boundary conditions.

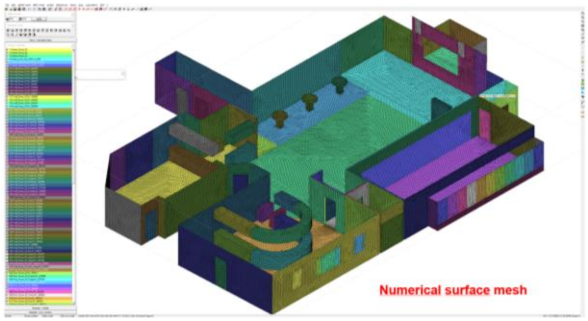
FlogoCFD: parallel solver for unsteady, turbulent incompressible flows enhanced with heat and mass transfer/buoyancy effects.

Numerical method characteristics:

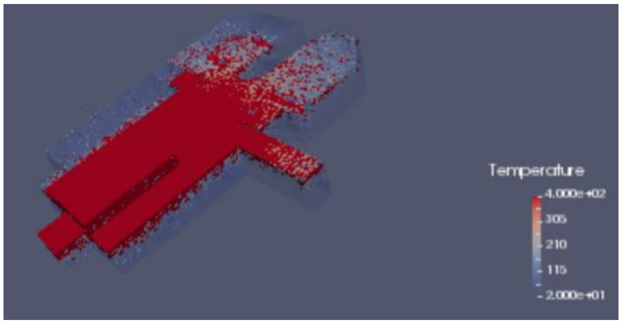
- Flow and heat transfer equations are solved strongly coupled.
- It is based on the pseudocompressibility concept for the solution of steady/unsteady, 2D/3D, turbulent flows
- It uses hybrid unstructured meshes.
- Dual time stepping: Second order three point backward scheme in physical time, first order backward Euler implicit scheme in pseudotime. Local time stepping in pseudotime for convergence acceleration.
- Upwind scheme for inviscid fluxes (second or third order accurate for mean flow and first order accurate for turbulence model)
- Central scheme for viscous fluxes (second order accurate)

Turbulence Models

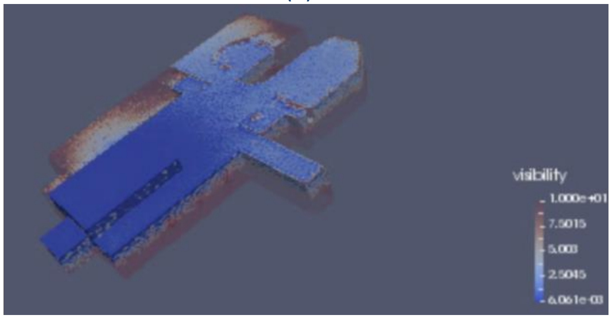
- Standard k-ε turbulence model
- k-ω SST
- k-ω TNT
- k-ε MMK
- Large Eddy Simulation (LES)



(a)

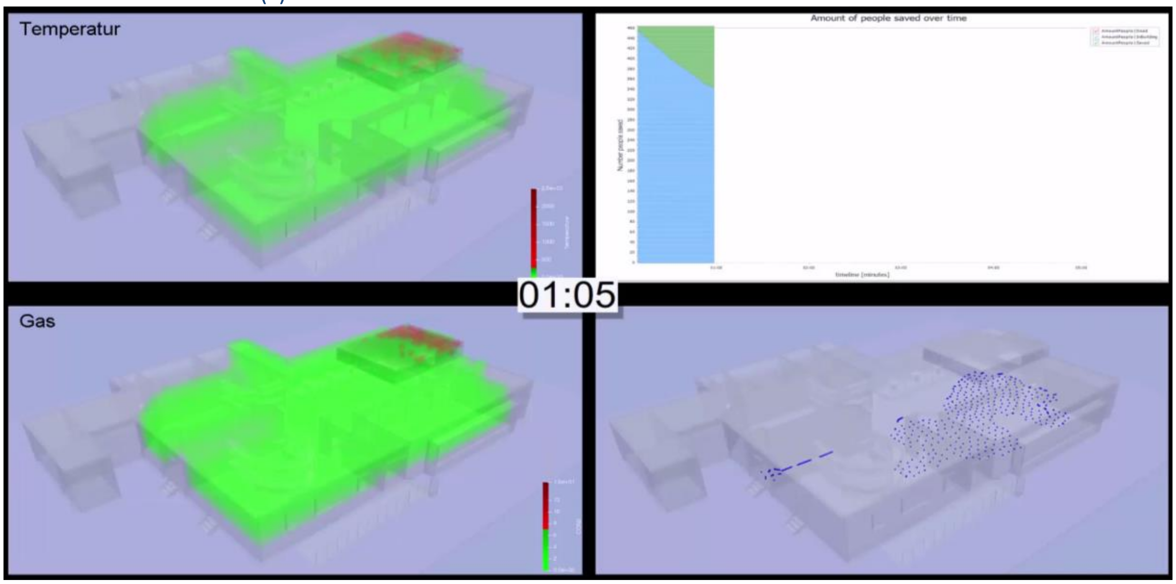


(b)



(c)

__Figure 1:__ Indoor fire simulation.
(a) Numerical surface mesh in FiDES modeler
(b) Temperature evolution
(c) Visibility evolution and
(d) Synchronized fire-crowd simulations



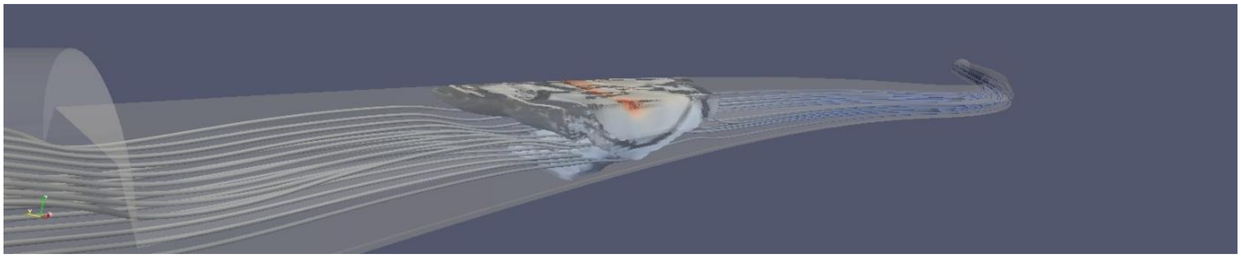
(d)

Implementation

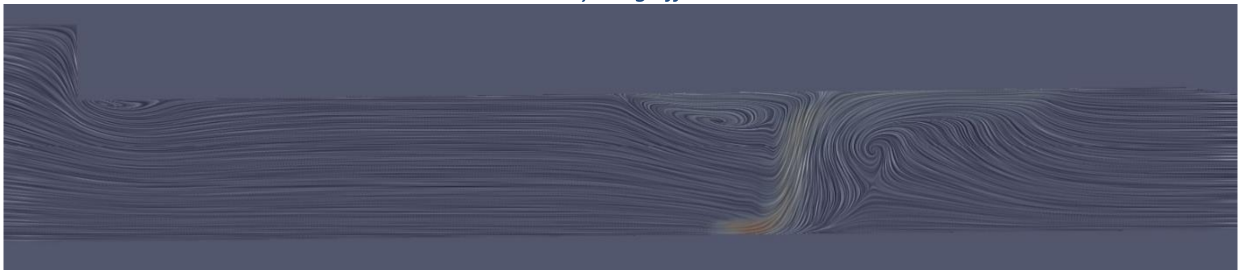
- Parallel CFD code based on the MPI protocol (MPICH2 and OpenMPI 64 bit implementations). The Single Program Multiple Data (SPMD) approach has been adopted. Mesh partitioning in subdomains is required to assign each processor/node a subdomain.
- Applied on hybrid unstructured meshes.
- Radiation model based on Finite Volume Method.
- Contaminant concentration field (soot, combustion products).
- Fire suppression by sprinklers (Lagrangian treatment and droplets' tracing).
- Simulation of HVAC systems.
- 1D simplified heat conduction through walls coupled with the CFD simulation.

New Capabilities

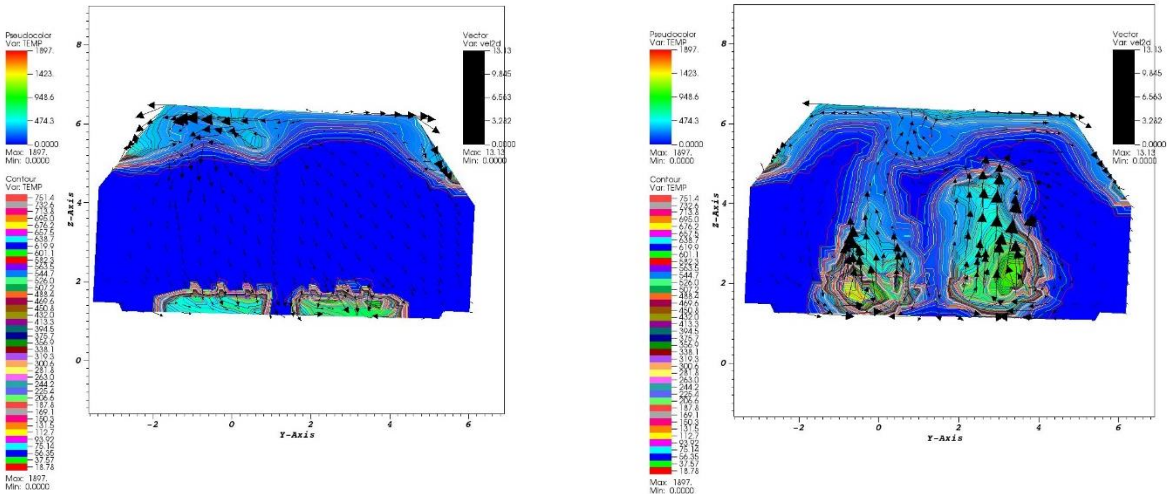
- Model generation from IFC (IFC4) files. Recognition of spaces, simplification of geometry and imposition of boundary conditions.
- Advanced Finite Volume model for heat radiation
- Heat Release Rate Properties of different materials
- Implementation of dynamic geometrical elements
- Incorporation of HVAC and BACS
- Modelling of active and passive fire protection systems
- Modeling of toxic gas dispersion considering additional buoyancy effects
- Coupling with crowd simulation application for evaluating evacuation strategies via a TCP/IP communication model
- Participation of smoke in the modeling of thermal radiation
- Material library with suitable properties
- Fire spread on the lining and the furniture.
- Evaluation of the results according to human safety criteria. Definition of suitable metrics/KPIs



Backlayering effect



Streamlines

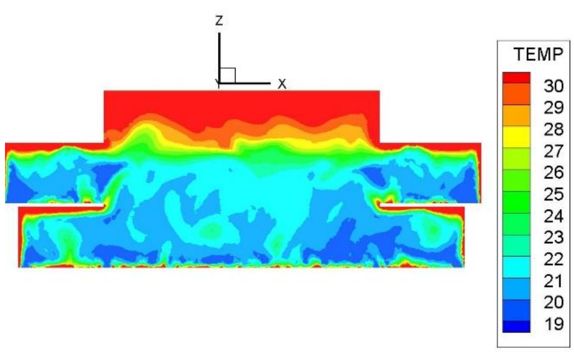


Velocity/temperature field on a tunnel cross section

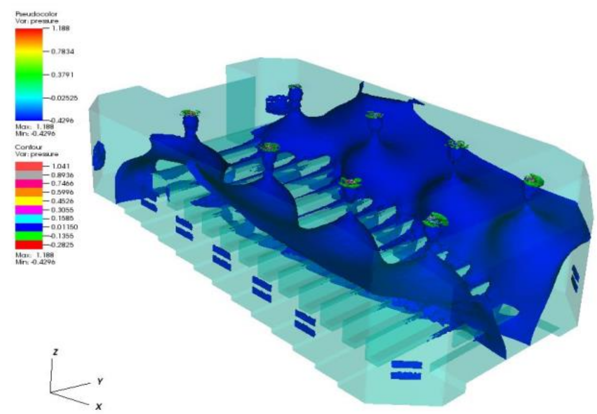
***Figure 2**: Simulation of fire events in tunnels*

Areas of Application

- Home Land Security Sector for the simulation of fire and toxic gas dispersion events (Figure 1a-c). Evaluation of CFD results according to human safety factors.
- Coupling with crowd simulation application (Figure 1d).
- Simulation of fire spread in tunnels. Dimensioning of tunnel lining subject to thermal loads (Figure 2).
- Estimation of thermal loads and thermal resistance of structures.
- Indoor climate prediction with emphasis to thermal comfort (Figure 3).
- Architectural resign of building or building blocks in urban environment for harnessing wind flow for natural ventilation purposes (Figure 4).
- Automatic exchange of data between CFD and Structural analysis application.



Displacement cooling/ventilation



Mixing cooling in an amphitheater

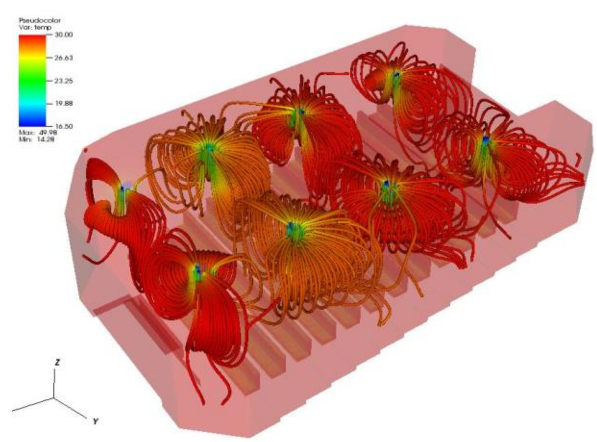
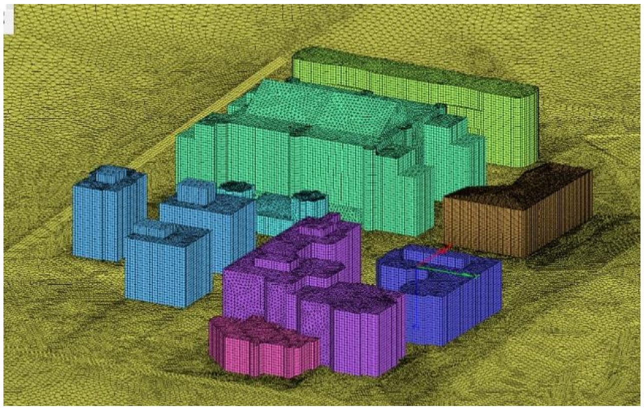
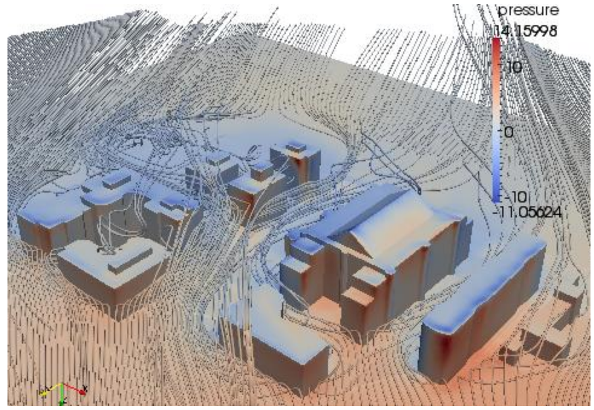


Figure 3: *CFD simulation for indoor microclimate. Thermal comfort and HVAC performance assessment simulations.*

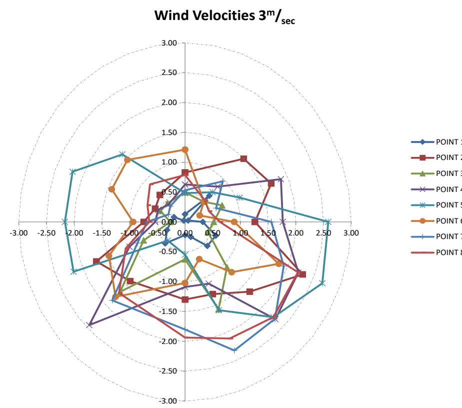


Numerical surface mesh



Streamlines, pressure distribution on buildings' surfaces

Figure 4: *CFD simulation for outdoor wind field analysis. Wind potential for natural ventilation & pedestrian comfort estimation.*



Wind velocity around the building of interest