# Appendix II

# Work done at the Christian Doppler Laboratory for Applied Computational Fluid Dynamics (CD)

### *Mont Blanc tunnel case study*

As a first test case for the a-priori generation of CFD and combustion data the Mont Blanc tunnel geometry was selected. A drawing of the tunnel cross section is depicted in fig. 1.

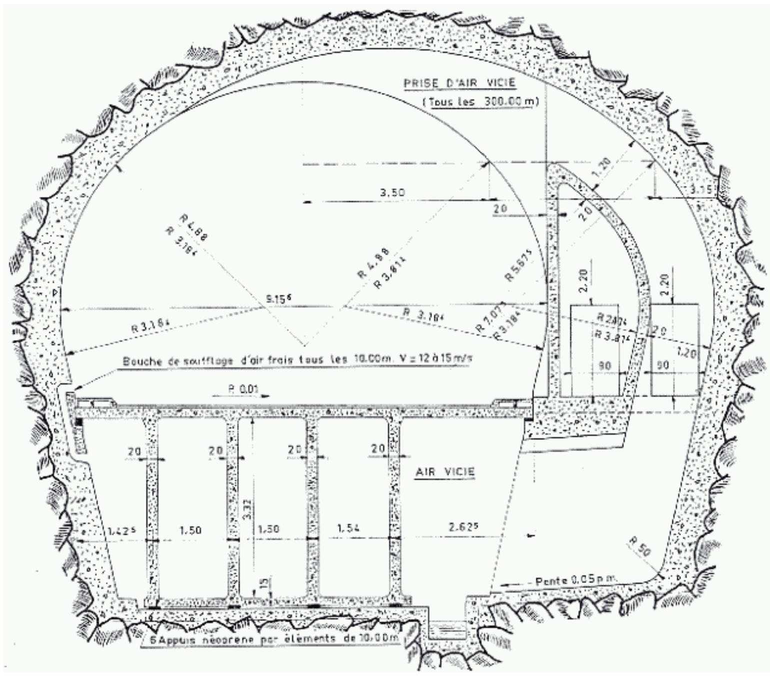



**Figure 1:** *Mont Blanc tunnel cross section*

Since a flow and combustion simulation based on the entire tunnel length of 8 [km] is out of the scope of today's computers, a characteristic section of 150 [m] containing 15 fresh air inlet openings and one exhaust air outlet located midways was chosen. This section was discretised into 130.000 control volumes. For parametric studies of fire hazards, modelling of a section of this length was considered to be sufficient.

A pressure difference between the two sides of the tunnel section has been applied corresponding to the pressure differences between the portals due to barometric reasons and wind pressure. This pressure difference resulted in an axial tunnel ventilation speed of up to 4 [m/sec].

A fresh air ventilation rate of 75 [m³/sec] was supplied by every ventilation section. The amount of fresh air was equally distributed via inlet openings located at a 10 [m] distance between each other at one bottom side of the tunnel. The supply of fresh and the removal of exhaust air takes place via channels located below the road surface, as can be seen in fig. 1.

For the first calculations a pool fire heat source delivering an energy amount of about 20 [MW] has been modelled. In this context it has to be mentioned that the heat load generated by a truck being on fire is usually in the range of 30 [MW], the one of a passenger car fire is approximately 5 [MW]. Therefore the assumption of a heat source of 20 [MW] is quite representative. Only the smoke spread has been computed in this first simulation and therefore no combustion and radiation model has been taken into account.

For the given heat load it can be seen that under the assumed hazard scenario the ventilation system is not able to remove the hot combustion gases through a single exhaust air opening (located in the middle of the computational model). Furthermore a significant stratification of the hot gases located under the tunnel ceiling can be observed. The simulation results can be observed in figs. 2a-2d**.**

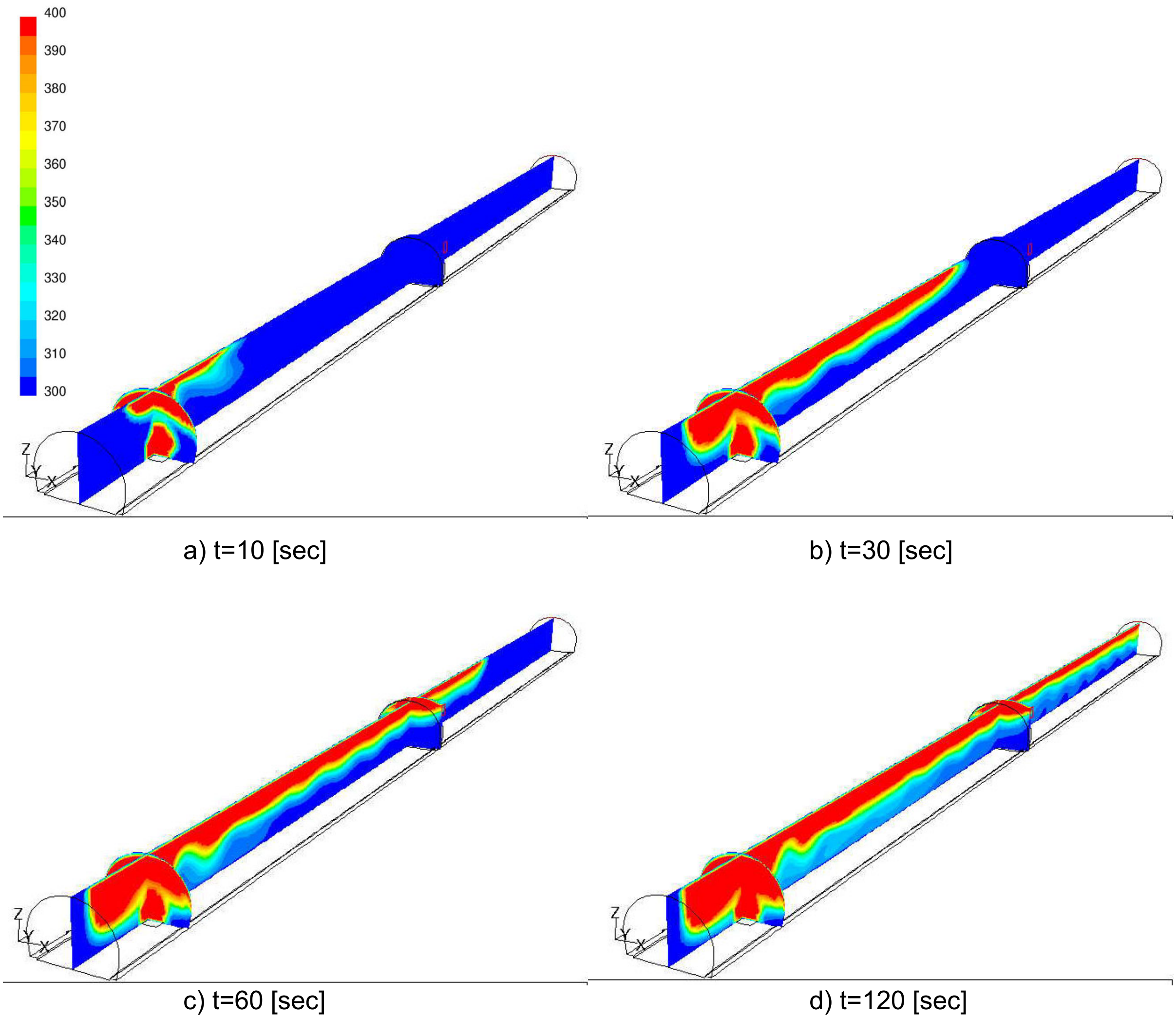



**Figure 2:** *Mont Blanc tunnel case study. Temperature distribution at different times after onset of 20 [MW] heat load.*

### *Gleinalm tunnel case study*

As a second test case a hazard scenario inside a section of the Gleinalm tunnel in Styria, Austria, was considered. A drawing of the Gleinalm tunnel cross section is depicted in fig. 3.

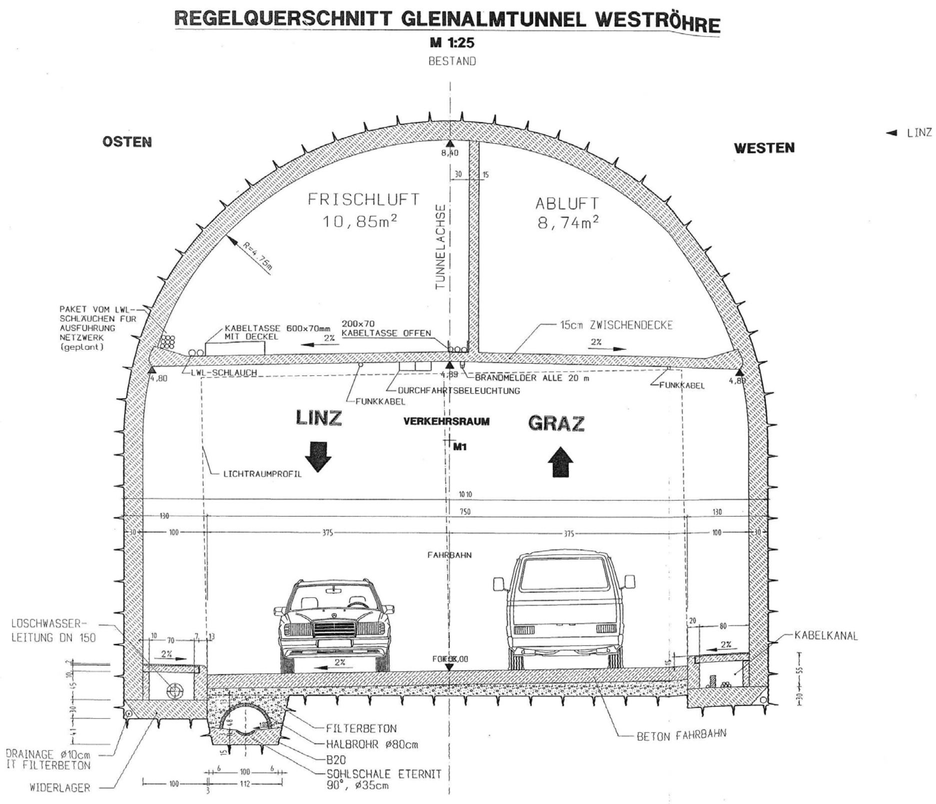
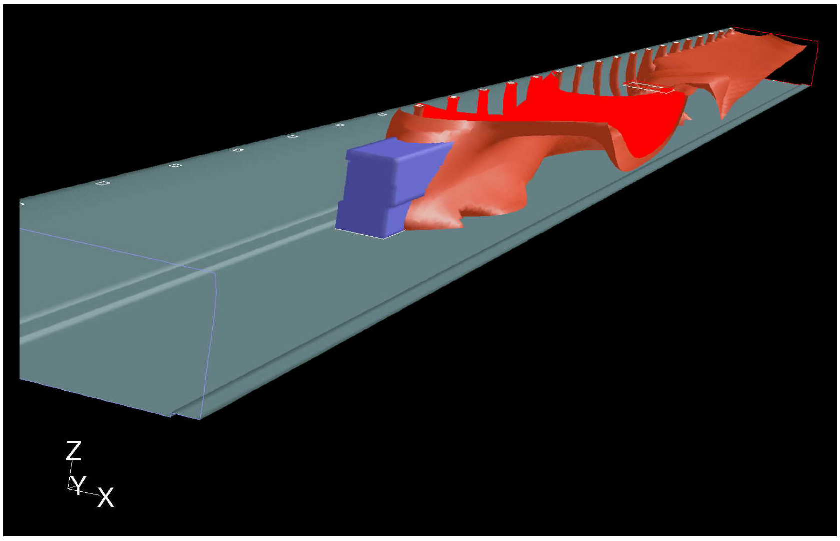



**Figure 3:** *Gleinalm tunnel*

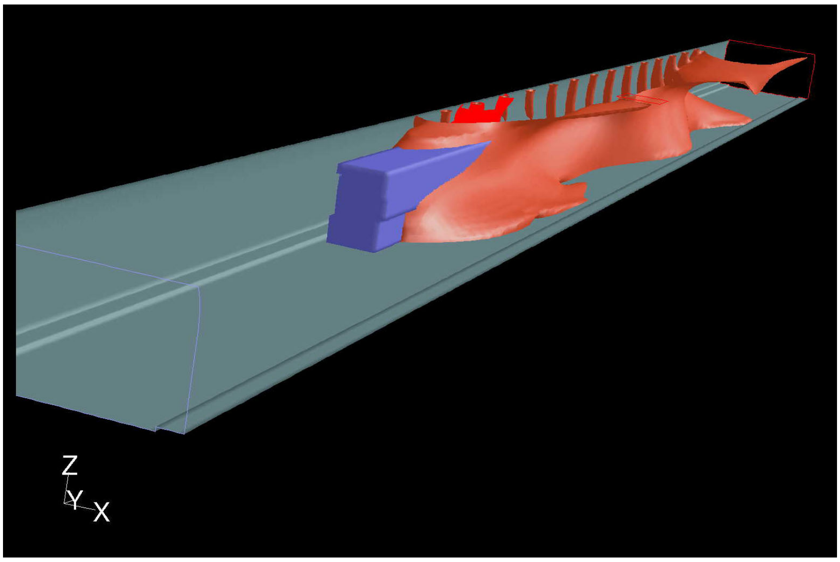
The Gleinalm tunnel was of particular interest since just recently a new evacuation system consisting of updated telephones, rescue cabins, light signs, etc. was installed. The exhaust air openings containing novel blinds have been increased to 3x3 [m] in size. These blinds are to be opened in the immediate neighbourhood of the fire origin and should be closed everywhere else. In case of a hazard running the ventilation system on full load, smoke should be removed in the immediate neighbourhood of the fire and no toxic combustion gases should spread below the ceiling and eventually (after having cooled down) sink towards fire escaping people.

As in the Mont Blanc tunnel case only a characteristic section of 150 [m] length was considered for the simulation study. Again a pressure difference between both sides of the tunnel section was applied. The fresh air inlets are now located at the ceiling of the tunnel every 6 [m]. Fresh and exhaust air are ventilated through channels located above the ceiling of the tunnel traffic enclosure.

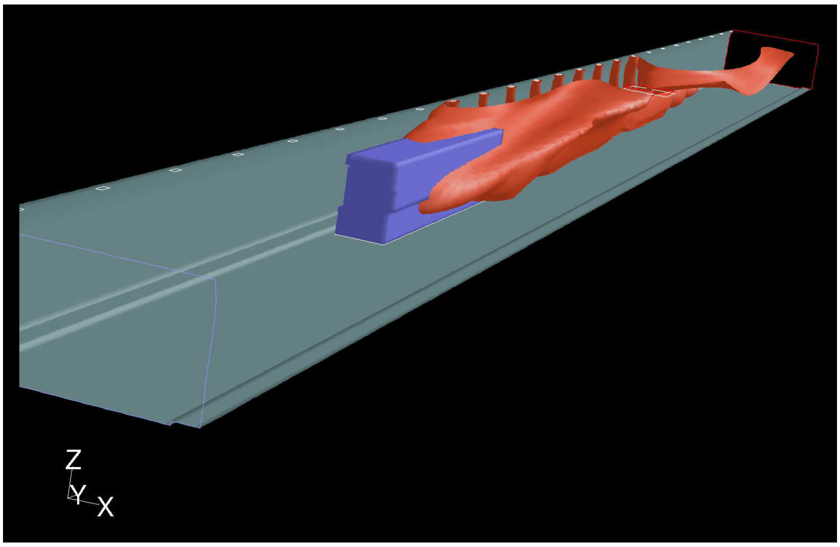
A heat load of 30 [MW] corresponding to a truck fire was applied and the computed smoke spread at different time levels for an iso-surface value of temperature of 400 K are depicted in figs. 4a-4c. For the given ventilation rate of 100 m³/sec it is evident that not all smoke spreading along the ceiling can be removed by the ventilation system through one single exhaust air opening.




a) t=30 [sec]




b) t=60 [sec]




c) t=120 [sec]

**Figure 4:** *Computed smoke spread around a truck at different time levels. Movement of a temperature iso-surface of 400 K is displayed.*

***Assessment of combustion model***

First results of a 2D-version of the Lattice-Boltzmann server including the above mentioned combustion model are shown in fig. 5. In this case a truck fire in an axially ventilated tunnel was modelled.

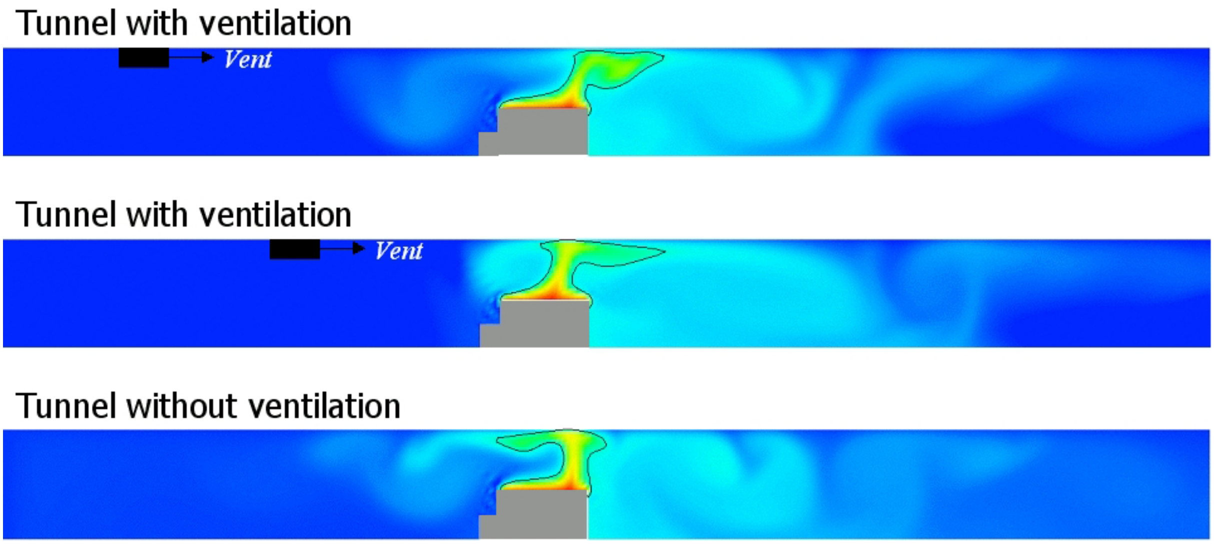



**Figure 5:** *2D Simulation of a Tunnel Fire Temperature Distribution*

These two dimensional simulations served to test the combustion model. As a result of all the work undertaken so far it can be concluded that the proposed model is working properly and real time flow and combustion simulations seem to be feasible. Therefore at this point in time no major risks in achieving the simulation targets of the project can be identified.

## Literature

[1] Smagorinsky J., B. Galperin and S.A. Orszag, Eds.
*“Some Historical Remarks on the Use of Non-linear Viscosities.*
*In Large Eddy Simulation of Complex Engineering and Geophysical Flows.”*
Cambridge University Press, 3-36