Abstract

Deployment of mobile fans has a big impact in helping to confine the smoke in case of fire in road tunnels. With doing so, increasing the level of security for both drivers and fire fighters. This thesis puts the focus on investigating the performance of mobile fans in a tunnel by means of flow field measurements, a 1D network model and CFD simulations. The mobile fan used for the experiments is a BIG MGV-L105 fan and the experiments were conducted in the Jan De Vostunnel in Belgium. In order to get a correct measurement of the velocity profile in the tunnel a framework was made based on the codes NBN EN 12599 and NBN EN ISO 5802. The framework allows for a detailed comparison with CFD simulations on top of the current comparison of the average velocity for network models. A literature study pointed out that both codes are complementary to each other.

A feasibility study was performed to investigate the performance of the mobile fans in case of fire in a tunnel. Three performance criteria were set: reach a velocity equal or larger than the critical velocity, allow a backlayering length of 30m or 50m. The nine cases considered were a combination of a certain HRR (5MW, 30 MW or 100MW) and the mobile fan set-up(one L105 fan, two L105 fans and one L105 plus one L125 fan). The feasibility study revealed that for an incident tunnel, with one mobile fan and a fire of 5MW, the situation was feasible with an allowance of a limited backlayering length. Wall roughness calculations highlighted the importance of performing pressure measurements over a length L in a zone with an expected constant pressure drop. Especially for tunnels with an irregular cross-section.

Comparison of the measured data with the simulated results lead to several propositions to alter the boundary conditions and thus creating a more accurate simulation. These adaptations involved implementing the louvered vents and changing the length and the number of the eddies, together with changing the root mean square (RMS) velocity fluctuation.