VENTILATION DESIGN TOOLS AND VALIDATION
Sturm P.J., Bacher M., Schmölzer G., Beyer M.
Graz University of Technology, Austria

ABSTRACT
The ventilation system of a tunnel has to fulfil two requirements. It has to ensure that the in-tunnel air quality is good enough to allow safe passage through the tunnel. In addition it has to improve the self rescue conditions in case of a fire. To achieve these objectives specific design tools have to be used. These tools might be simple or complex, but they must be appropriate and accurate enough to ensure sound design of the system. Very often simple tools are employed for complex problems and complex tools are used for simple installations. This paper looks at different tool applications and checks validity against measured data.

Keywords: ventilation design, design tools, measurements

1. INTRODUCTION
The design of tunnel ventilation has to focus on two objectives. The ventilation system has to be capable of providing sufficient fresh air for the tunnel and must be able to control the air flow in the case of a fire.

For the first purpose it is necessary to know the amount of pollutants emitted by the vehicles. For this emission factors have to be used to describe the emission behaviour of a vehicle during its tunnel passage. Such factors are defined in national and international guidelines. However, as these factors try to describe the average emission behaviour under certain tunnel specific situations and the “average” vehicle in a tunnel is subject to permanent change in its emission behaviour, the factors must be verified on an almost regularly basis. Ventilation fans must provide enough power to deliver the air volumes needed to dilute pollution and ensure that concentration and visibility values remain within acceptable levels.

The second purpose is to control the air flow within the tunnel according to a predefined ventilation scenario. Hence the ventilation power has to be big enough to overcome all the pressure losses which occur within a tunnel and its ventilation system. There are many ways to calculate head losses. Depending on the complexity of the system the methods and tools used for calculating the pressure losses might range from a simple summing up of individual head losses to complex 3D CFD tools. Common to all of these tools is that they need assumptions, boundary conditions, simplified geometry etc. Hence, validation against measured data is required in order to establish tool performance.

2. VENTILATION DESIGN TOOLS
In Austria, ventilation design is based on the guidelines RVS 09.02.31 and 09.02.32. These guidelines state the requirements for normal and incident ventilation. Normal ventilation control can be based either on in-tunnel air quality (visibility or CO concentration) or on the number of vehicles passing through the tunnel. Correct incident ventilation involves the provision of a predefined air flow with a defined velocity of the smoke/air mass at the incident location. Hence tools are needed in order to estimate the fresh air amount for normal operation as well as to match ventilation according to the needs of operation in an incident case. Within this paper the focus is put on ventilation design in general and incident ventilation in particular.
2.1. Calculation tools for ventilation power

As soon as the fresh air demand is fixed the ventilation can be designed for normal operation. For the fire case the specifications concerning air/smoke volumes and air velocities inside the tunnel are defined in the various regulations. In Austria this is done in the RVS 09.02.31, on the international level PIARC 2008 gives a good overview of the worldwide situation. The given volume flow multiplied by the pressure loss (and an efficiency coefficient) results in the ventilation power.

2.1.1. Simple approach (sum of pressure losses)

The objective is to calculate the pressure difference on the basis of a summation of pressure losses due to resistances within the tunnel (and ducts in the case of a transverse ventilation system), atmospheric pressure differences between the portals, the influence of moving/stopped vehicles, and thermic influences. Various literatures describe specific methodology (e.g. Freibauer 1978), and resistance coefficients and friction numbers can be found in Idelchik and Fried 1989. The approach allows for a quick and simple calculation of fan power needed to move the air through the tunnel or the respective ventilation ducts. However, this method depends strongly on (well tested) empirical factors and as in general tunnel geometry often deviates from standard configurations a certain element of uncertainty remains. Thus, the usage of such factors – especially for transverse ventilated tunnels – needs some experience.

A drawback of the above method is that only stationary situations can be considered with a sufficient degree of accuracy.

2.1.2. One dimensional calculation scheme

The main interest in aerodynamic flows concerns the flow in the direction of the tunnel. Hence, normally only flow information along the longitudinal axis of the tunnel is of interest. The calculation is based on solving the conservation equations for mass (for air as well as water), energy, momentum and a passive scalar in one dimension. To do this the tunnel is split into a number of cells in which the respective equations are solved. In order to allocate the pressure correctly in the momentum equation a staggered grid is used. An explicit scheme is used for solving the time dependency in the case of a transient application.

Heat transfer between the hot air/smoke mass and the surface by forced convection followed by heat conduction into the walls is calculated on the basis of the energy conservation equation.

The discretisation of the equations is based on a finite volume scheme, the numerical solution on a TDMA solver.

The model used at our lab allows the implementation of individual fans in longitudinally ventilated tunnels, various splits of the flow due to on/off ramps and a combination of tunnel geometry and ventilation ducts. The calculations are performed for moist air. This allows for the consideration of water vapour e.g. due to water mist systems.

2.1.3. Three dimensional calculation scheme

The basis of 3D models is more or less the same as for the 1D approach described above. The relevant conservation equations are solved in three dimensional space. The partial differential equations for momentum, energy and scalar are transformed into a system of linear equations and solved in an iterative process. The closure of the Reynolds Averaged Navier Stokes equations (RANS) is done using a turbulence model. The models applied for the calculations shown below are mainly FLUENT or CFX from the ANSYS group (www.ansys.com). The advantage of 3D models is the possibility to have a very good geometrical representation of the part of the tunnel under consideration. But as they need a lot of memory and calculation...
time only small parts of a tunnel can be calculated within reasonable calculation times. It needs to be noted, however, that in most parts of the tunnel 3D information is not of interest. Only a segment with complex flow situations due to complex geometry of other purposes (fire, air injection or extraction, etc.) needs to be handled with a 3D model. Nevertheless, a sound description of the boundary conditions as well as a proper selection of models and model parameters is required in order to end up with reliable results.

3. TRANSVERSE VENTILATION – AIR INJECTION

Air injection by Saccardo type nozzles are nowadays frequently used in order to avoid installation of jet fans inside the tunnel or in order to use the capabilities of existing air supply in transverse ventilated tunnels for smoke control in an incident case.

3.1. Measurements in the Katschberg tunnel

In the Katschberg tunnel Saccardo type fresh air injection nozzles are employed for controlling the airflow and confining the smoke in case of a fire. As in such cases air is injected into the tunnel room at a high velocity, a complex 3 dimensional behaviour of air flow can be expected. Figure 1 shows a sketch of the tunnel section with the injection nozzle and the measurement arrangement. Volume flows in the duct, as well as those on both sides of the nozzle inside the traffic room were measured. Furthermore, the pressure difference between the undisturbed flow inside the duct and the two measurement cross sections for volume flow inside the traffic room were recorded. Volume flows were monitored employing ultrasonic devices whereas the pressure differences were recorded with differential pressure capsules.

Figure 1: Sketch of the air injection nozzle and the location of the performed measurements

The velocity measurement inside the air supply duct was performed with a 3D point monitor (plane 1 in Figure 1). In order to get the appropriate flow field a correction according to Pokorny et al (1981) was performed. The velocity measurements at the cross sections 2a and 2b were based on path averaged velocity information. No corrections had to be applied. The pressure difference measurements were installed such as to give the static pressure. Total pressure was calculated as sum of static and dynamic pressure.

The geometry of the injection damper is known. Knowledge of the opening angle of the damper (90° represents fully open) allows for calculation of the open damper cross section.
and hence the injection velocity. Based on the total pressure loss and the velocity in the nozzle it was possible to derive resistance coefficients for the injection nozzle. For opening positions 0° - 90° the pressure loss between pos. 1 and 2a was calculated, for angles > 90° the difference was taken between 1 and 2b (always in direction of the main flow). The axial fan was used at two different operating points, n1 with 370 rpm and n2 with 742 rpm.

**Table 1:** Results of the measurements of the air injection nozzle

<table>
<thead>
<tr>
<th>rpm [min⁻¹]</th>
<th>position [degree]</th>
<th>Air flow [m³/s]</th>
<th>Diff. pressure Δp stat [Pa]</th>
<th>Δp tot [Pa] (damper)</th>
<th>cross-section* [m²]</th>
<th>uₘ [m/s]</th>
<th>ζ measured ** [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>n₂ 138</td>
<td>179.98</td>
<td>1283.11</td>
<td>1375.12</td>
<td>5.55</td>
<td>32.44</td>
<td>2.22</td>
<td></td>
</tr>
<tr>
<td>n₂ 138</td>
<td>185.51</td>
<td>1270.07</td>
<td>1368.60</td>
<td>5.55</td>
<td>33.44</td>
<td>2.07</td>
<td></td>
</tr>
<tr>
<td>n₂ 50</td>
<td>211.53</td>
<td>934.86</td>
<td>1082.21</td>
<td>6.35</td>
<td>33.31</td>
<td>1.54</td>
<td></td>
</tr>
<tr>
<td>n₂ 35</td>
<td>176.73</td>
<td>1178.77</td>
<td>1269.87</td>
<td>4.75</td>
<td>37.17</td>
<td>1.56</td>
<td></td>
</tr>
<tr>
<td>n₂ 18</td>
<td>108.11</td>
<td>1498.41</td>
<td>1517.25</td>
<td>2.56</td>
<td>42.20</td>
<td>1.53</td>
<td></td>
</tr>
<tr>
<td>n₁ 138</td>
<td>105.96</td>
<td>337.69</td>
<td>364.46</td>
<td>5.55</td>
<td>19.10</td>
<td>1.69</td>
<td></td>
</tr>
<tr>
<td>n₁ 125</td>
<td>116.88</td>
<td>355.02</td>
<td>398.27</td>
<td>6.79</td>
<td>17.21</td>
<td>2.18</td>
<td></td>
</tr>
<tr>
<td>n₁ 50</td>
<td>124.26</td>
<td>332.38</td>
<td>382.60</td>
<td>6.35</td>
<td>19.57</td>
<td>1.58</td>
<td></td>
</tr>
<tr>
<td>n₁ 35</td>
<td>106.48</td>
<td>333.93</td>
<td>361.18</td>
<td>4.75</td>
<td>22.39</td>
<td>1.21</td>
<td></td>
</tr>
<tr>
<td>n₁ 18</td>
<td>77.13</td>
<td>372.37</td>
<td>378.89</td>
<td>2.56</td>
<td>30.11</td>
<td>0.75</td>
<td></td>
</tr>
</tbody>
</table>

* flow-through area of the damper, completely opened (90°) 8.29 m²
** Mean air density over measuring period 1.0973 kg/m³

### 3.2. Model validation

#### 3.2.1. Simple approach

The simple approach is based on the calculation of the resistances and the consequent pressure losses (see section 2.1.1). In order to model the air injection geometry has to be simplified somewhat. The whole path from the air supply duct to the traffic room is split into 4 sections. Section 1 covers the 90° bending of the air when being diverted from the air supply duct into the damper. Section 2 covers the entrance into the damper, while section 3 represents the damper itself. Section 4 stands for the merging of two flows, that from the damper and the existing one in the traffic room. Figure 2 depicts the simplified model for this specific section between the air supply duct and the traffic room. The resistance coefficients were taken according to Idelchick and Fried (1989), adapted to the geometry of this section.

**Table 2:** Resistance coefficients, comparison between selected and measured values

<table>
<thead>
<tr>
<th>rev. *</th>
<th>n₁</th>
<th>n₁</th>
<th>n₁</th>
<th>n₁</th>
<th>n₂</th>
<th>n₂</th>
<th>n₂</th>
<th>n₂</th>
<th>n₂</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle [°]</td>
<td>138</td>
<td>125</td>
<td>50</td>
<td>35</td>
<td>18</td>
<td>138</td>
<td>138</td>
<td>50</td>
<td>35</td>
</tr>
<tr>
<td>ζ total</td>
<td>2.03</td>
<td>2.13</td>
<td>1.75</td>
<td>1.66</td>
<td>1.60</td>
<td>2.06</td>
<td>2.06</td>
<td>1.78</td>
<td>1.66</td>
</tr>
<tr>
<td>ζ measured</td>
<td>1.69</td>
<td>2.18</td>
<td>1.58</td>
<td>1.21</td>
<td>0.75</td>
<td>2.22</td>
<td>2.07</td>
<td>1.54</td>
<td>1.56</td>
</tr>
</tbody>
</table>

5th International Conference ‘Tunnel Safety and Ventilation’ 2010, Graz
Except for one test at n1 and an 18° opening angle, the differences between simulation and observation are within 10% to 15%, with simulation values always being higher than measurement values.

3.2.2. CFD Calculations

The CFD model represents a tunnel section with a length of 350 m and a tunnel cross section of 48 m² and an air duct area of 9 m², and is similar to that described in section 2.1.3. An additional cubic zone at the tunnel portal was introduced to impose atmospheric pressure outlet conditions. The damper was placed in the middle of the tunnel geometry and had a free flow cross section of 8.3 m² (at a damper flaps angle of 90°). The calculation was performed by using a hybrid mesh with approximately two million elements. This mesh consists of tetrahedral elements in the area with the damper and of wedge elements in the remaining domain. Figure 3 shows the CFD geometry used for this calculation.

The boundary condition for the air duct was a constant mass flow inlet based on an air density of 1.08 kg/m³. For the tunnel outlet conditions, a relatively static pressure of 0 Pa was set on the vertical plane of the added cubic zone. Additionally a wall roughness for the tunnel wall and for the damper was included. The turbulence was simulated with the standard k-ε model, and a logarithmic wall function was applied.

Figure 4 depicts the flow pattern in the symmetrical plane. It can clearly be seen, that the supply air – coming through the duct– is reversed by the damper and results in a highly unsteady flow phenomena downstream of the damper. The exit velocity from the damper is some 38 m/s, the maximum speed at ground level reaches some 20 m/s.

Table 3 shows the comparison between measurement and simulation for three different cases. Notice that there is a small difference in the angles of the damper flaps between simulation and measurement. This arose because the simulations were performed during the design phase, while the measurements were performed at the point of best operation. In most cases the volume flow was higher in the tests than in the simulation. However, the pressure losses showed a comparable result.
Figure 3: CFD geometry of a part of the calculation domain

Figure 4: velocity contours in the symmetry plane for calculation case 2 (see Table 3)

Table 3: Comparison between measurements and CFD calculation

<table>
<thead>
<tr>
<th></th>
<th>Case 1</th>
<th></th>
<th>Case 2</th>
<th></th>
<th>Case 3</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle (flaps))</td>
<td>°</td>
<td></td>
<td>°</td>
<td></td>
<td>°</td>
<td></td>
</tr>
<tr>
<td>Volume flow</td>
<td>m³/s</td>
<td></td>
<td>m³/s</td>
<td></td>
<td>m³/s</td>
<td></td>
</tr>
<tr>
<td>Dp-stat.</td>
<td>Pa</td>
<td></td>
<td>Pa</td>
<td></td>
<td>Pa</td>
<td></td>
</tr>
<tr>
<td>Dp- tot.</td>
<td>Pa</td>
<td></td>
<td>Pa</td>
<td></td>
<td>Pa</td>
<td></td>
</tr>
</tbody>
</table>

4. AIR FLOW IN COMPLEX STRUCTURES

Wherever the presence of physical constraints increases ventilation duct complexity simple calculation schemes may no longer be appropriate. The reason is that the flows upstream as well as downstream of the elements under consideration are not at all uniform (over the cross section). Hence the assumptions on which the simple resistance coefficients are based are not valid. The following example is based on a ventilation section of the cavern section of the Tauern tunnel (Sturm 2009).
As part of the upgrading of the ventilation of the Tauern tunnel the number of the fresh air supply fans was reduced from two to one per ventilation section. Thus it was possible to remove the exhaust air fan from an elevated location and place it on the same platform as the fresh air supply fans. This was necessary in order to avoid free hanging stainless steel exhaust ducts which would have had to be insulated to protect against excess radiation heat in the case of fire.

Figure 5 shows the CAD model for the two exhaust air fans and ducts supplying the two central ventilation sections. As can be seen, multiple 90° elbows are lined up within a short distance. This results in a non-uniform flow pattern upstream of each of the elbows (with guiding blades) and hence in high head losses. In addition, the regions with very low velocities, or even backflow in the corners, are quite big (Figure 7). Taking the whole section between the exit of the axial fans and the entrance into the vertical shaft into account the total pressure drop accounted on average over the cross sections to 335 Pa, based on the CFD calculations (Figure 7). For the same region a simple calculation based on resistance coefficients (Idelchik 1989) was performed. Using the same volume flows the pressure losses summed to 250 Pa.

As technical problems arose, the measurements needed for validation of calculated results are still pending.
5. CONCLUSION

The design of a ventilation system is based on all available information as well as on the usage of certain design tools. However, uncertainties still have to be taken into account. The final construction of the ventilation ducts, or installations such as guiding blades, are never as they appear in the CAD drawings. This is due to construction and installation tolerances and unforeseen changes needed during construction. Such situations can only be handled by applying additional safety margins in the calculation.

The use of calculation tools entails a different approach. A tunnel and its ducts can in most cases be considered as a one dimensional system. The main flow is along the longitudinal axes, secondary flows in the cross section play a minor role. Hence a quite simple approach of assuming resistance coefficients for the single parts of the tunnel-duct system and summing up the head losses of all these sections is in many situations sufficient. However, there are certain situations where the main assumptions for the usage of simple tools are not valid as the behaviour of the flow in such stretches is multidimensional. For such situations more complex tools such as CFD need to be used. However, the usage of CFD tools requires detailed knowledge of the principles of numerical flow calculations (usage of turbulence models, grid independency etc.). Otherwise the results gained by CFD might be more incorrect than those gained by the usage of simpler tools.

6. REFERENCES

RVS 09.02.31; Basic Principles, Ventilation, Tunnel Equipment, Tunnel; FSV Wien, 2008

RVS 09.02.32; Fresh Air Demand, Ventilation, Tunnel; FSV Wien, 2009


PIARC 2008; Road Tunnels: Operational Strategies for Emergency Ventilation (in press)

Freibauer B. 1978; Bemessungsgrundlagen für die Lüftung von Straßentunneln, Straßenforschung Heft 87, Bundesministerium für Bauten und Technik, Wien, 1978

