Validation of air ventilation in tunnels, using experiments and computational fluid dynamics

T Myrvang*, H Khawaja
Department of Engineering and Safety, UiT-The Arctic University of Norway

ABSTRACT
The study presented in this paper concerns the ventilation system inside tunnels. The ventilation system is responsible for the removal of exhaust gases produced by vehicles and for providing a clear view throughout the tunnels in routine operations and in the event of fire. The ventilation system has several stages, which are equipped with one or more fans and can be activated together or separately. The objective of this study is to find a better correlation between the air velocity and number of ventilation stages inside a tunnel. A small experimental model representing a miniature model of a tunnel was built for the study. In addition, the problem was modelled using computational fluid dynamics (CFD) in ANSYS® Fluent 18.0 for comparison and verification. The results from experiments with the miniature model and simulations from the CFD study were found to be in good agreement and in relation to a real-case scenario. The results also indicated that an increase in the active number of ventilation stages does not result in a linear increase in the air velocity inside the tunnel.

1. INTRODUCTION

Tunnels in Norway are administrated by the Norwegian Public Roads Administration (NPRA). The NPRA is divided into five different geographical regions, each of which has the responsibility for the road network within that region [1]. Tunnels with a length over 1000 meters and an average daily traffic volume of over 1000 are equipped with a ventilation system consisting of one or more groups of fans [2].

The ventilation system is responsible for the removal of exhaust gases produced by vehicles and for providing a clear view throughout the tunnels in routine operations and in the event of fire [3]. These tunnels are also equipped with sensors for measuring the concentration of nitrogen dioxide (NO_2), nitrous oxide (NO) and carbon monoxide (CO) [4]. Both NO_2 and CO are highly toxic, and it is crucial that the ventilation system effectively removes these gases and ensures a safe passage through the tunnels.

In order to ensure that the level of toxic gases in the tunnels is low, ventilation stages are used, where the polluted air is diluted and transported out of the tunnel in the same direction as the traffic [5]. The ventilation stages can be equipped with one or more fans which can be activated together or separately. In such systems, ventilation stages are activated based on the concentration level of one of the exhaust gases, as given in Table 1 [6].

*Corresponding Author: tomyrvang@gmail.com

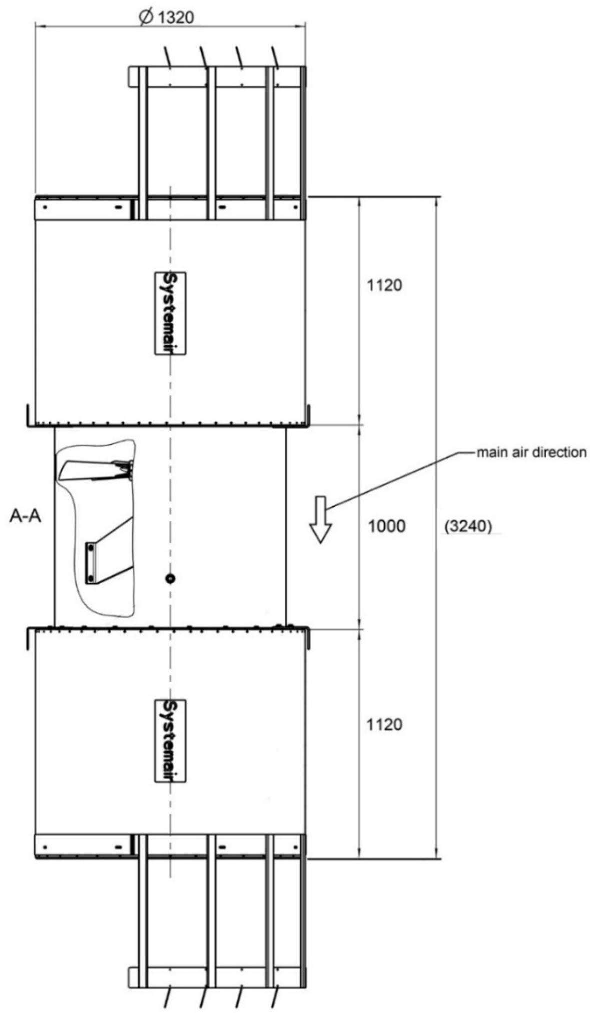
Figure 1: *Lærdalstunnelen* in Norway, the longest road tunnel in the world (24.51 km). Photo: Jørn Eriksson, CC BY 2.0.

Table 1: Concentration of NO_2, NO and CO in ppm to activate the ventilation stage(s) in the tunnel [6].

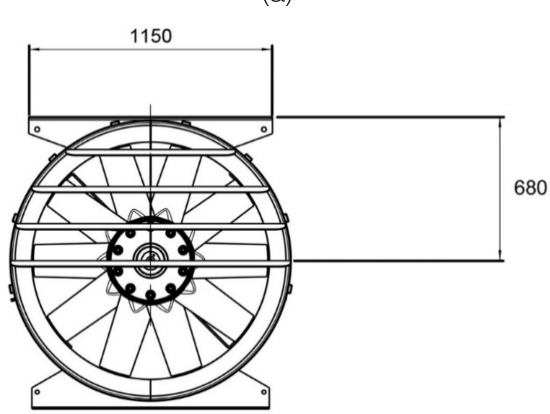
Ventilation stage(s)	Emergency exit located at 350 – 500 m from the tunnel entrance (ppm)			Hard shoulder located in the middle of the tunnel (ppm)		
	Nitrogen dioxide (NO_2)	Nitrous oxide (NO)	Carbon monoxide (CO)	Nitrogen dioxide (NO_2)	Nitrous oxide (NO)	Carbon monoxide (CO)
1	0.8	5.0	20	0.4	4.5	10
2	1.0	6.5	30	0.5	5.1	15
3	1.2	8.0	40	0.6	5.7	20
4	1.4	9.5	50	0.7	6.3	25
Alarm	1.5	10	100	0.75	6.8	50

Table 1 shows data for an up to four-stage ventilation system. Tunnels normally use one to four stages, at which fans are respectively activated at various power settings. For instance, stage one refers to a power setting of 25 %, stage two refers to a power setting of 50 %, stage three to 75 %, and stage four to 100 % of the fans. NPRA [7] uses a series of impulse jet fans of the type AJ 1120 for tunnels that require ventilation [6]. A technical drawing of AJ 1120 is shown in Figure 1, and technical data is provided in Table 2.

These fans are placed in groups, each normally consisting of four to six fans. The fans are grouped together to reduce the voltage drop from the supply cables and where each group shares the same electrical system. Multiple groups are placed to provide redundancy in the case of failure in the group and where the number of fan groups depends on the length of the tunnel. [7]


Ø 1320
1120
Systemair
main air direction
A-A
1000
(3240)
1120
Systemair

(a)


1150
680

(b)

(c)

Figure 2: The figure axial fan AJ 1120 (a) Side view (b) Front view (c) Installed in the tunnel *Sørkjostunnelen*, Sørkjosen Norway [8]. Given dimensions are in [mm].

Figure 3: Picture showing the entrance for the *Tromsøysundtunnelen* in Tromsø, Norway.

Table 2:Technical data of a single AJ 1120 fan [8]

Static thrust in both directions	$\approx 980\ N$
Air volume	$\approx 28.4\ m^3/s$
Outlet velocity	$\approx 28.8\ m/s$
Designed air density	$\approx 1.2 kg/m^3$

The Norwegian Public Roads Administration has a standard for describing the shapes of the cross-section areas of tunnels, referred to as tunnel profiles. The most common tunnel profile in the northern region of Norway is called T9.5 [7]. This profile describes tunnels with a two-lane road, a total width of 9.5 meters, a road width of 7 meters and a standard cross-section area of 52.6 m2. The speed limit in these tunnels normally ranges from 60 km/h to 80 km/h [6]. Figure 3 shows the entrance of the *Tromsøysundtunnelen* in Tromsø, Norway, which has a T9.5 tunnel profile. This tunnel is equipped with AJ 1120 fans in various ventilation stages to keep the tunnel safe for traffic.

One of the challenges that has been encountered is that of understanding the correlation between the activated ventilation stages and their impact on ventilation. The given study addresses this by utilizing a state-of-the-art computational fluid dynamics (CFD) model of a tunnel with various ventilation stages. CFD has been used to study tunnels for various scenarios [9-14]. The problem is also addressed through empirical analysis of a miniature model of a tunnel.

2. METHODOLOGY

The objective of this study is to find a better correlation between the air velocity and number of ventilation stages inside a tunnel. A small experimental model representing a miniature model of a tunnel was built for the study. In addition, the problem was modelled using computational fluid dynamics (CFD) in ANSYS® Fluent 18.0 [15] for comparison and verification.

2.1. Experimental model

The experimental model was used both to investigate the correlation between air velocity and number of ventilation stages and to test different fan operations. The model consists of four 1.2-metre in length and 315 mm in diameter spirally wound aluminum pipes and nine 40 mm in diameter PC fans. The PC fans represent a small-scale model of the axial jet fan used inside the tunnels. The dimensions of the pipe were proportional to the T9.5 tunnel. In this case, diameter of the circular pipe is compared with the hydraulic diameter of a semi-circular cross-section of T9.5 tunnel and found to be in a ratio (i.e. 0.315 m:11.57 m = 1:37). Similarly, PC were chosen to be as close in dimensional proportionality to AJ 1120 axial fans (i.e. 0.04 m:1.32 m = 1:33). The positions of PC fans were also chosen to be proportional to the positions of the fans in T9.5 tunnel (see Figure 6). This case study represents the flow in the turbulent regime with large value of relative pipe roughness hence does not need Reynolds number to be same.

Figure 5 shows the inside of the experimental model and how the PC fans are mounted onto the pipes and aligned with the use of threaded rods. For the experimental model, the PC fans are controlled using a control system designed for this study, enabling the fans to be controlled via a PC.

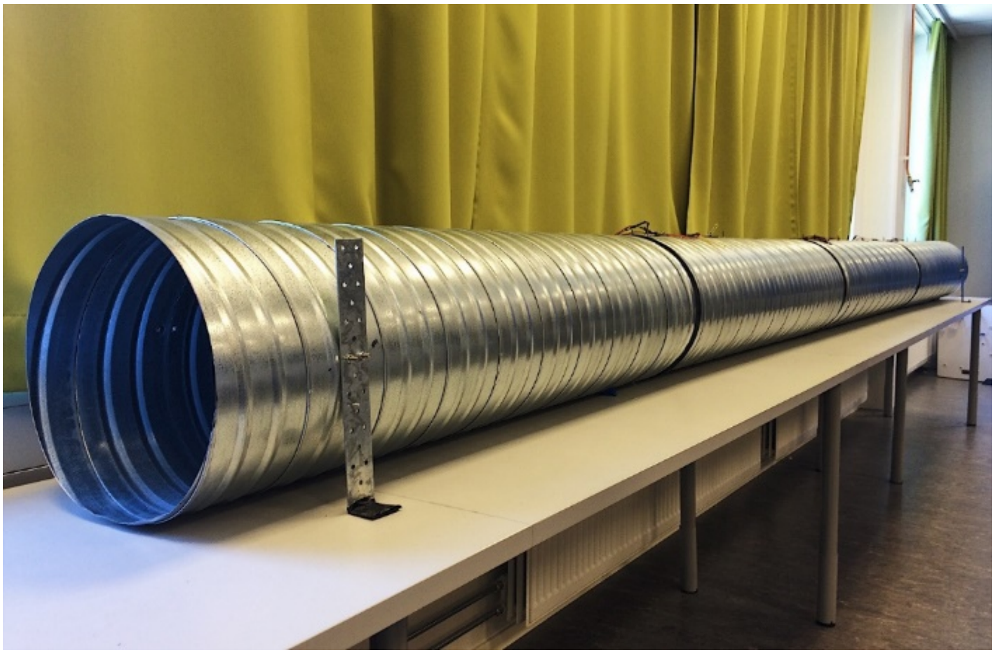


Figure 4: Picture showing the experimental model use for the study. This miniature model represents a straight section of a real tunnel

Figure 5: Photograph showing the inside of the experimental model and how the PC fans are mounted

Figure 6 shows the overall length and diameter of the experimental model and the placement of the fans. The diameter of the PC fans (40 mm) is in accordance with the diameter of the experimental model (315 mm) is proportional to the diameter of the AJ 1120 fans (Figure 2b) installed in a tunnel with tunnel profile T9.5. Figure 7 shows the markings of the airflow inlet, outlet, longitudinal measurement points (from one to eight), and three lateral points (from A to C).

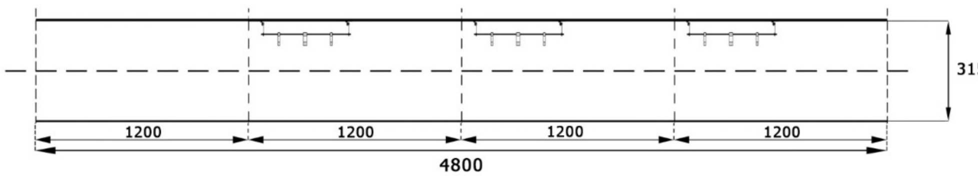


Figure 6: Diagram showing overall length and diameter of the experimental model. It also shows the length of each of the pipes and the placement of the fans in relation to each other. Given dimensions are in [mm].

In this study, nine PC fans are activated sequentially, starting with the fan closest to the inlet. For the nine cases, the air velocity was measured in eight different longitudinal and three lateral locations. This resulted in 24 different points, which were then averaged for each of the nine cases. This produced measurement for the average air velocity inside the experimental model was used for comparing the simulation data.

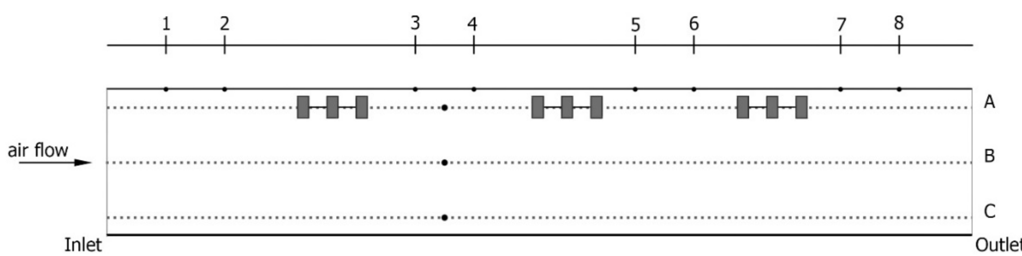


Figure 7: Diagram showing where measurements in the experimental model were made, the inlet and outlet of the model and flow direction

2.2 Simulation model

The simulation model is used to analyze how the air develops inside the tunnel from the tunnel inlet to the tunnel outlet by activating a given number of modelled fans.

Figure 8 shows a technical drawing of the cross-section of the simulation model, which represents a section of a real tunnel and is modelled after tunnel profile T9.5 and the dimension for the axial jet fans.

The simulation model consists of six modelled fans; the placement of the fans is approximately the same as in a real tunnel. The whole model is 800 metres in length, and the fans are placed at 250, 400 and 550 metres from the tunnel inlet, as shown in Figure 9. Figure 10 shows a sketch of the simulation model and gives an overview of the fluid domain and the numbering of each of the fans, as well as their placement in relation to each other.

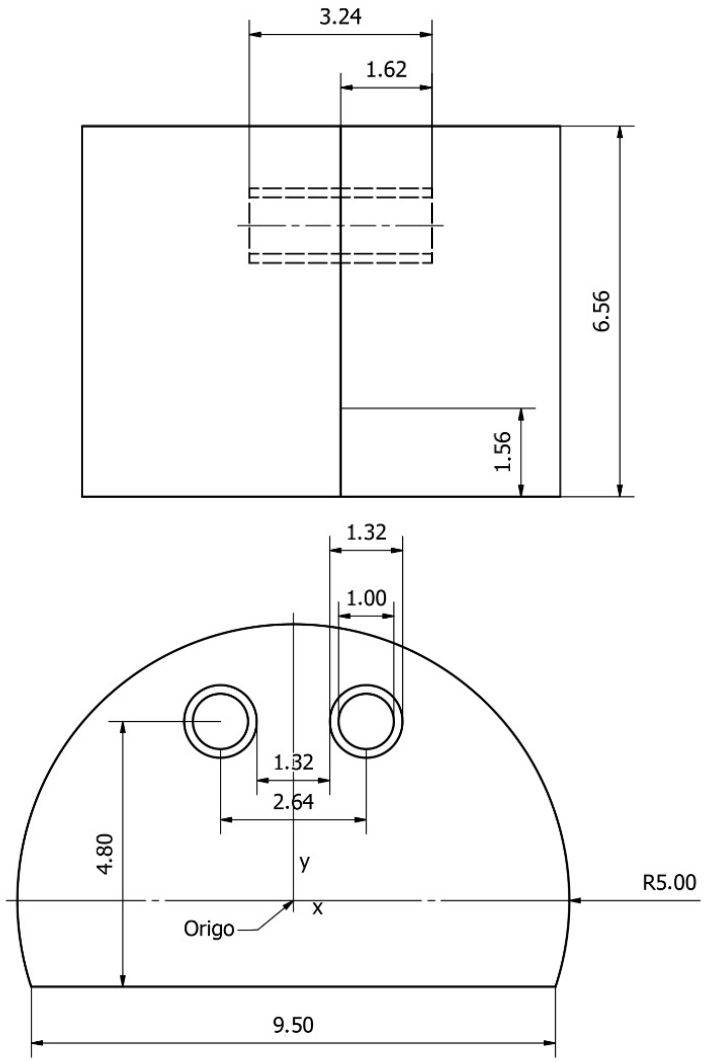


Figure 8: Technical drawing of the cross-section of the simulation model, also showing the dimension and placement of the fan model in relation to the cross-section. Given dimensions are in [m].

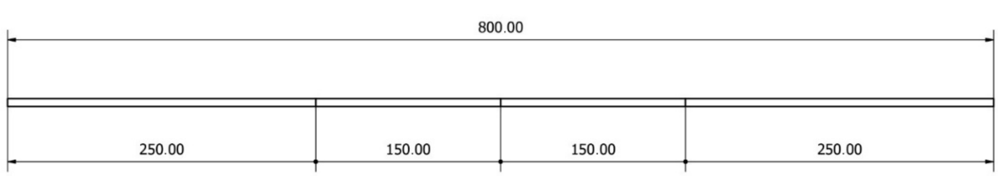


Figure 9: Diagram showing the overall length of the simulation model and the placement of the fans in relation to the simulation model. Given dimensions are in [m].

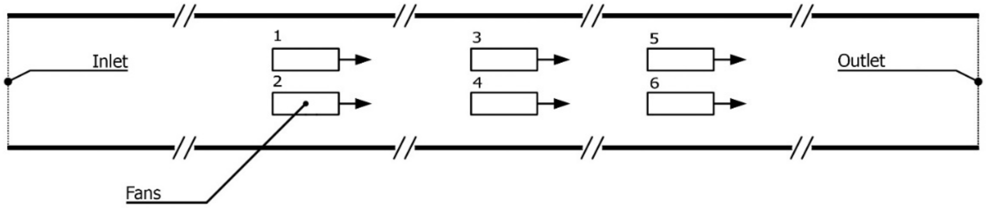


Figure 10: Sketch of the fluid domain of the simulation model. This top-down view shows the inlet and outlet of the model, the direction of the fluid flow and the numbering of the fans

For the simulation, ANSYS® Fluent [15] was used; the simulation model was built in Autodesk® inventor and imported to ANSYS® for further processing. For the turbulence model, the k-ε, Realizable, Scalable Wall Function was used. The simulation was set to pressure-based SIMPLE algorithm. For the simulation run, double precision variables were incorporated, and the solution was achieved using parallel processing. An auto-meshing function was used to generate the mesh and where Proximity and Curvature scaling was used. Mesh sensitivity analysis was performed to ensure the correctness of the results. Table 3 shows the specific parameters that were used for the final meshing of the CAD model. The final mesh grid for the CAD model consisted of 176,531 nodes and 505,000 elements.

Table 1: The parameters used for the meshing of the CAD model

Max Face Size	11.5m
Mesh de-featuring	yes
De-feature Size	5.75e-02m
Span Angle Center	Fine
Min Size	0.20m
Max Tet Size	18.0m
Proximity Min Size	0.1150m

For the boundary condition, the intake of the fans is defined as outlet and the exhaust of the fans is defined as inlet (Figure 13). Tunnel surface, road, and duct walls were defined as rough walls. The inlet and the outlet of the tunnel were selected as pressure inlet and outlet, respectively. The surfaces with applied boundary conditions are marked in Figure 14 and listed in Table 4.

Table 2: boundary condition for the different surfaces.

tunnel_inlet	Pressure-inlet
tunnel_outlet	Pressure-outlet
tunnel_wall	Wall
tunnel_road	Wall
DuctWalls	Wall
fan_inlet	Velocity-inlet
fan_outlet	Pressure-outlet

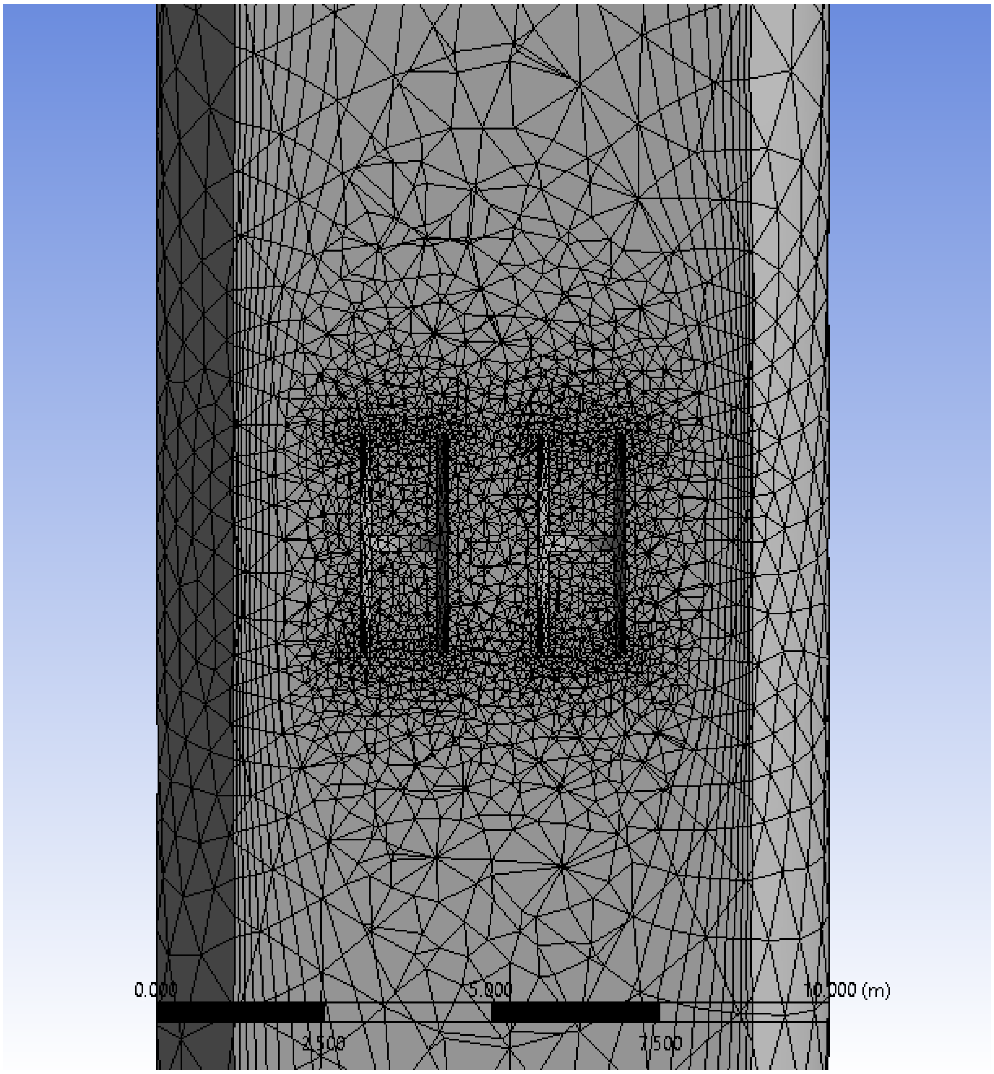


Figure 11: A top view of the generated mesh grid for the simulation model

The flow through the fans was defined as subsonic and with an initial value of 20.16 m/s. In Table 1, the outlet velocity is given as approximately 28.8 m/s. In [6], it is stated that 30 % of the thrust from the fans is lost due to part of the flow hitting the tunnel surfaces. This results in an outlet velocity of 20.16 m/s. The roughness for the tunnel wall has a value of 0.0025 m [6]. An estimated roughness for the tunnel road is 0.01 m.

For the CFD model, six different simulation runs were made (Table 5). The first run involved the activation of a single fan (Fan_1). In the second run, two fans were activated (Fan_1 and Fan_2). In the third run, three fans were activated (Fan_1, Fan_2 and Fan_3). Four fans were activated in the fourth run (Fan_1, Fan_2, Fan_3 and Fan_4) and five in the fifth (Fan_1, Fan_2, Fan_3, Fan_4, and Fan_5). In the final run, all six fans were activated (Fan_1, Fan_2, Fan_3, Fan_4, Fan_5 and Fan_6).

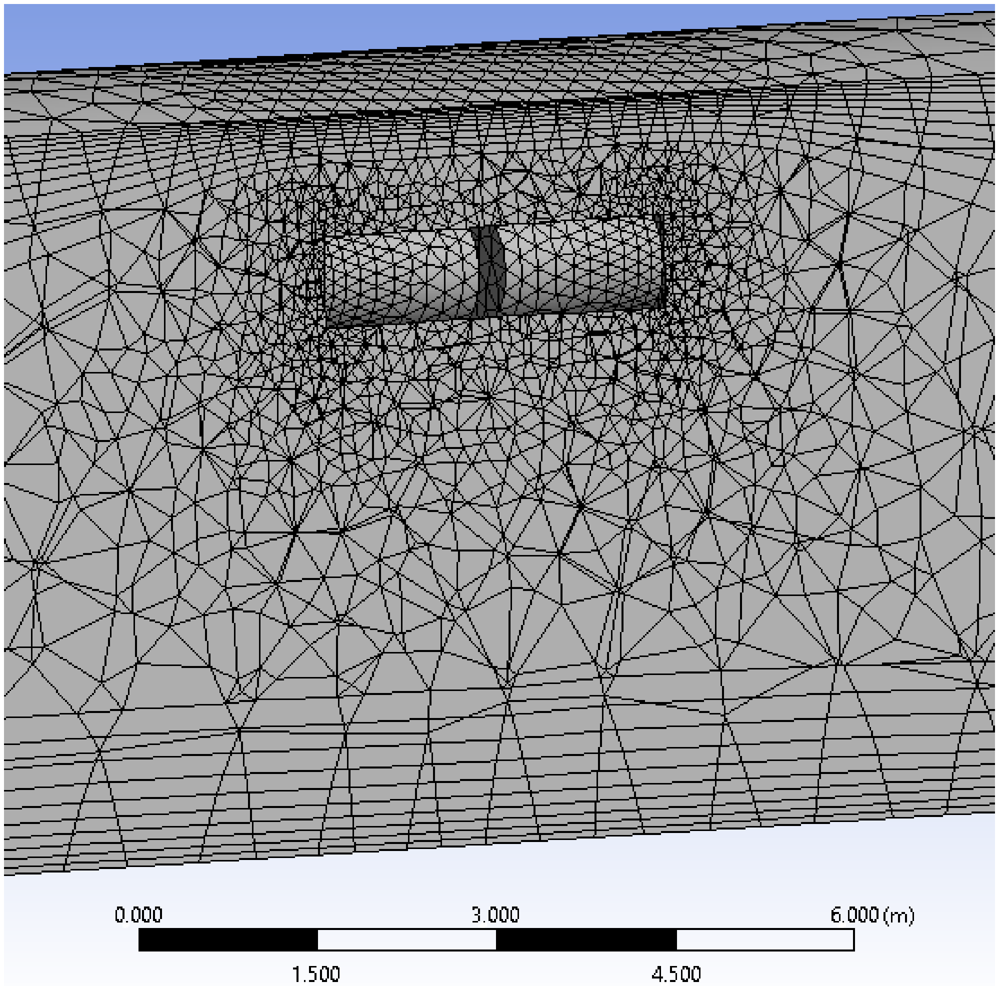


Figure 12: A side view of the generated mesh grid of the model. The final mesh grid for the simulation model consisted of 176,531 nodes and 505,000 elements

Table 5: Boundary conditions for the fan inlets for the different simulation runs.

Run 1	Fan_1	20.16 m/s
	Fan_2, Fan_3, Fan_4, Fan_5, Fan_6	0 m/s
Run 2	Fan_1, Fan_2	20.16 m/s
	Fan_3, Fan_4, Fan_5, Fan_6	0 m/s
Run 3	Fan_1, Fan_2, Fan_3	20.16 m/s
	Fan_4, Fan_5, Fan_6	0 m/s
Run 4	Fan_1, Fan_2, Fan_3, Fan_4	20.16 m/s
	Fan_5, Fan_6	0 m/s
Run 5	Fan_1, Fan_2, Fan_3, Fan_4, Fan_5	20.16 m/s
	Fan_6	0 m/s
Run 6	Fan_1, Fan_2, Fan_3, Fan_4, Fan_5, Fan_6	20.16 m/s

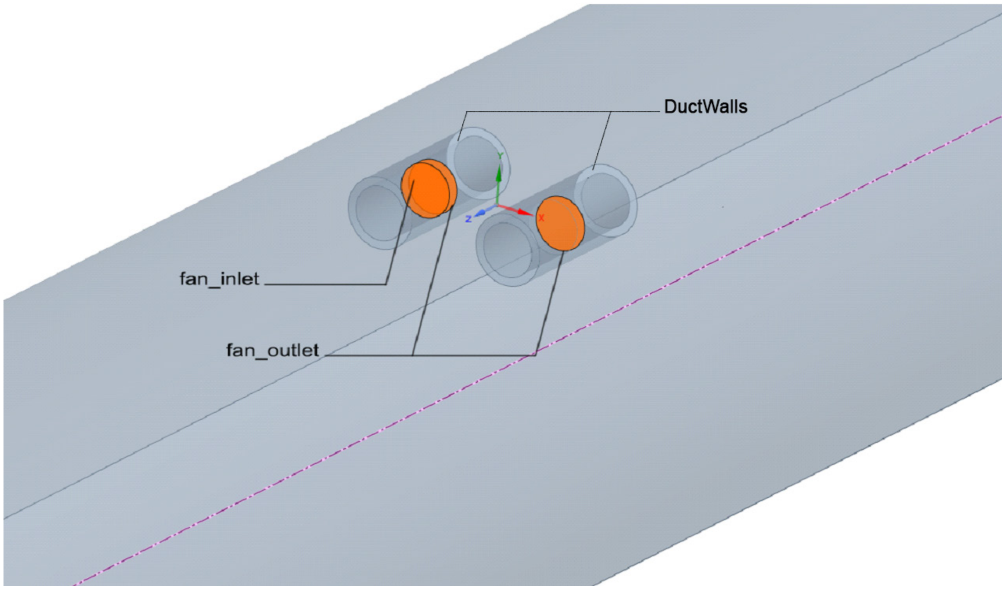


Figure 13: An overview of the model of the fans with the inlet and outlet to the fans highlighted

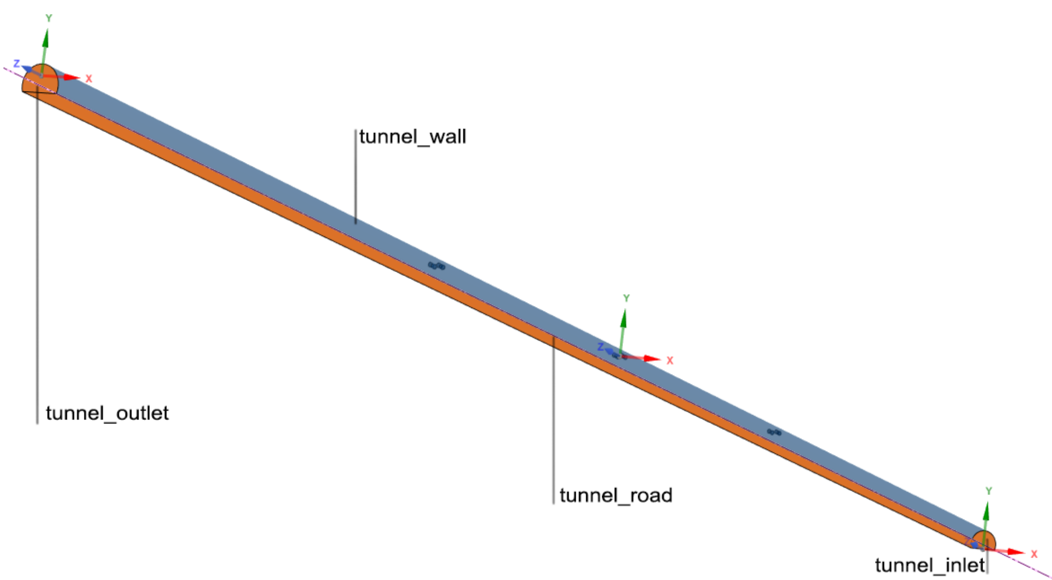


Figure 14: An overview of the CAD model with the inlet, outlet, road and tunnel wall highlighted

3. RESULTS AND DISCUSSIONS

This section discusses the results from the experimental and the CFD models. The presented results focus on the problem of investigating the relationship between the air velocity inside the tunnel and the number of fans activated.

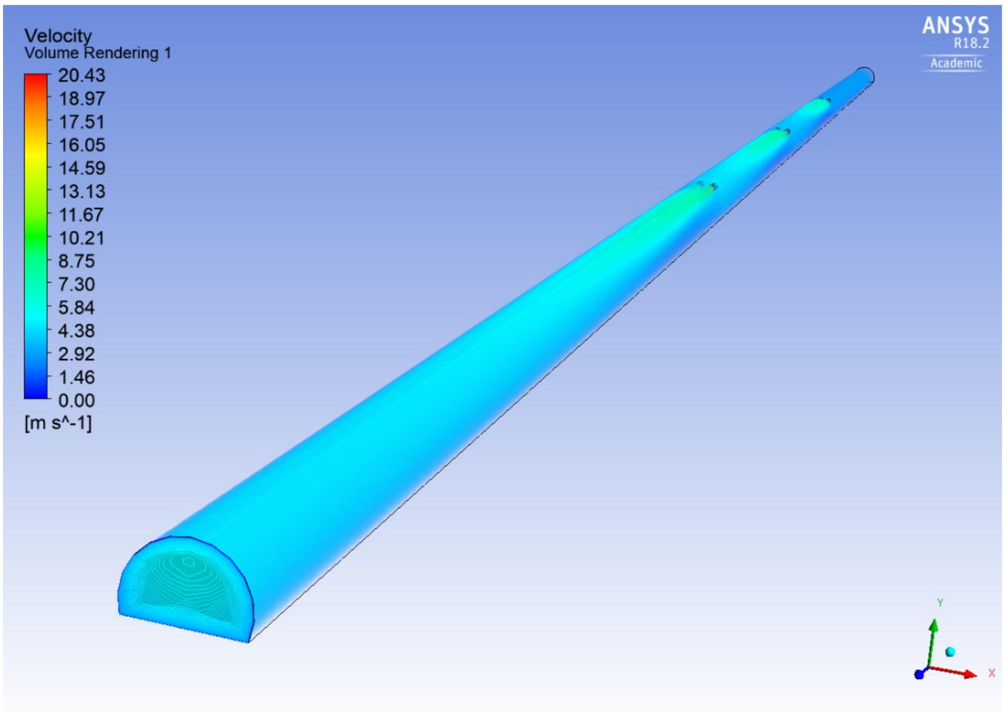


Figure 13: Visualization of the CFD result with all fans activated

Figure 15 shows a visualization of the simulation data from the CFD model, where all the fans are activated, by generating both volume rendering and a streamline plot of the air velocity.

Figure 16 shows the results from the experimental model. Air velocity in [m/s] is given on the y-axis and active number of fans [n] on the x-axis. The points show the average air velocity for one to nine fans in use. The two lines (purple and yellow) represent the extrema for the measurement and the value that the air velocity should have if there was a linear relationship.

For the CFD model, the data is presented in a similar way in Figure 17. As with the experimental model, air velocity in [m/s] is given on the y-axis and active number of fans [n] on the x-axis. The results from both the experimental model and the CFD model show that the air velocity dramatically increases when the first fans are activated. The air velocity from both Figures 16 and 17 also indicates that the effectiveness of each of the fans decreases when the number of fans activated increases. The results presented in Figures 16 and 17 also given in Tables 6 and 7, where data is converted into percentage values for comparison.

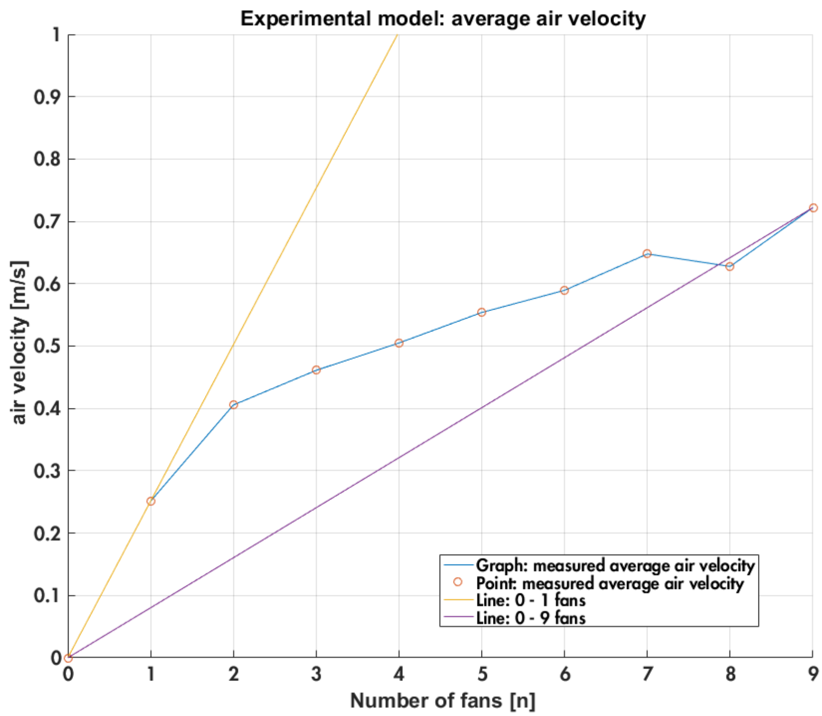


Figure 14: The result from the experimental model. The graph shows the average air velocity in correlation to number of fans active

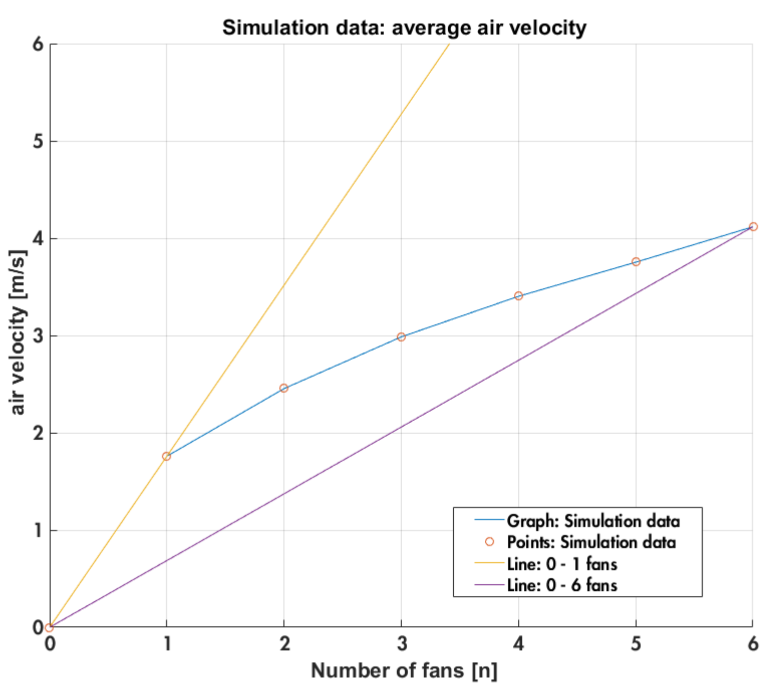


Figure 15: The figure shows the CFD result for the CAD model. The graph shows the average air velocity in correlation to number of fans active

Table 6: Data from experimental model

No. of Fans	No. of Fans (%)	Velocity (m/s)	Velocity (%)
0	0	0	0
1	11	0.25	34
2	22	0.4	55
3	33	0.46	64
4	44	0.5	68
5	56	0.55	77
6	67	0.59	81
7	78	0.65	89
8	89	0.63	87
9	100	0.72	100

Table 7: Data from the CFD model

No. of Fans	No. of Fans (%)	Velocity (m/s)	Velocity (%)
0	0	0.00	0
Run 1 (Table 5)	17	1.75	43
Run 2 (Table 5)	33	2.45	59
Run 3 (Table 5)	50	3.00	72
Run 4 (Table 5)	67	3.40	82
Run 5 (Table 5)	83	3.75	91
Run 6 (Table 5)	100	4.13	100

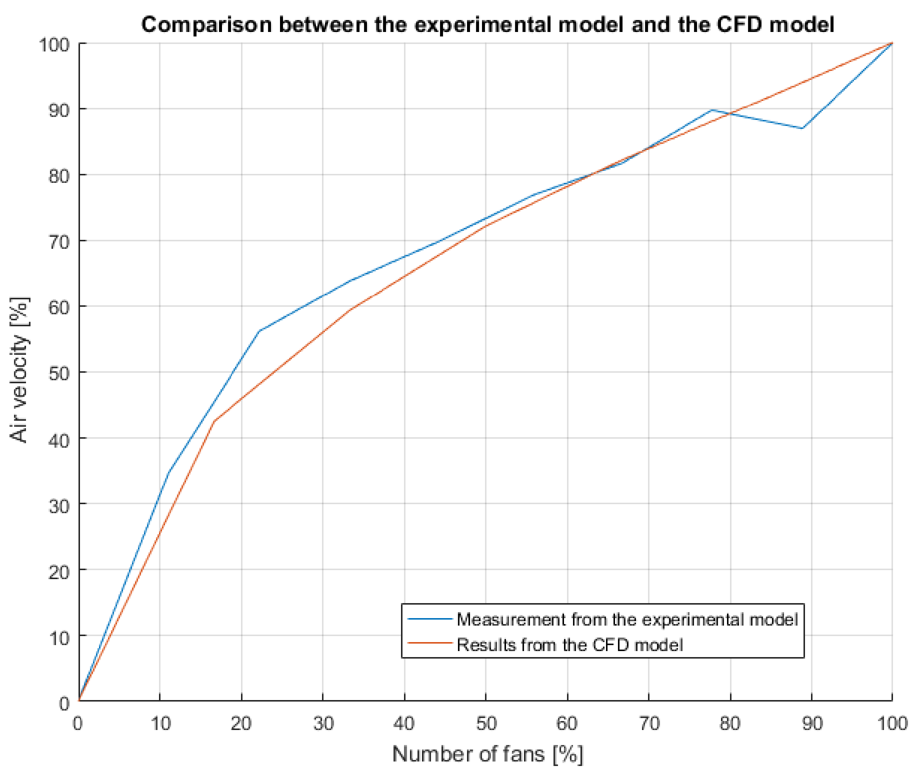


Figure 16: The figure shows a comparison between the experimental model and the CFD model

Figure 18 shows a comparison between the experimental model and the CFD model. Both the y- and x- axes are given a percentage, with the y-axis showing the air velocity and the x-axis showing the number of fans. From Tables 6, 7, and Figure 18, it can be observed that both models reach 50% to 60% of the produced air velocity when only 20% to 30% of the maximum number of fans are activated.

The results from Figure 16, Figure 17 and Figure 18 shows that both models follow a similar trend. The results also show that the increase is not linear with a decreasing gradient.

4. CONCLUSION

The following conclusions can be drawn from the given study:

1. A miniature experimental model was built to study the ventilation system of the tunnel. The flow characteristics were found to be relatable to those of a real-scale tunnel.
2. The CFD model of the tunnel was analyzed in ANSYS® 18.0 Fluent. The results were found to be relatable to those of the real-scale tunnels.
3. The experimental data from the miniature model and simulation data from the CFD study were compared and found to be in good agreement.
4. The results indicate that an increase in the active number of ventilation stages do not result in a linear increase in the airflow inside the tunnel. The trend was found to be a decreasing gradient and can be simply explained by exponential pressure losses with the increase in the flow velocity in the tunnels.

ACKNOWLEDGEMENTS

The authors acknowledge the support of Emmy Sortland, Emilia Kirkvik, and Jarle Johansen, from UiT-The Arctic University of Norway, and of Ole Jonny Varhaugvik from the Norwegian Public Road Administration (NPRA).

REFERENCES

[1] NPRA. Our tasks and roles (Original in Norwegian: Våre oppgaver og roller). 2018 [24-07-2018]; Available from: https://www.vegvesen.no/.

[2] NPRA, Handbook on Managing Tunnel V500 (Original in Norwegian: Tunnelveiledning Håndbok V500). 2016, Norwegian Public Roads Administration.

[3] Amundsen, F.H., The 5 major tunnel fires in Norway. 2017, Norwegian Public Roads Administration.

[4] Lotsberg, G., NO2/NOx volume ratio in three tunnels in Norway, Observations 2007-2013. 2013, Norwegian Public Roads Administration.

[5] Buvik, H. and R. Brandt, Long and steep tunnels - controlling fire ventilation (Original in Norwegian: Lange og bratte tunneler - styring av brannventilasjon). 2016, Norwegian Public Roads Administration.

[6] NPRA, Handbook on Managing Tunnel V520 (Original in Norwegian: Tunnelveiledning Håndbok V520). 2016, Norwegian Public Roads Administration.

[7] Varhaugvik, O.J., Technical queries about tunnels, T. Myrvang, Editor. 2017, Norwegian Public Roads Administration: Electro Division, NPRA northern region, Norway.

[8] Berg, S., Final documentation / FDV - Tunnel fans E6 Sørkjostunnelen (Original in Norwegian: Sluttdokumentasjon / FDV – Tunnelventilatorer E6 Sørkjostunnelen). 2017, Systemair AS.

[9] Middha, P. and O.R. Hansen, CFD simulation study to investigate the risk from hydrogen vehicles in tunnels. International Journal of Hydrogen Energy, 2009. 34(14): p. 5875-5886.

[10] Amouzandeh, A., M. Zeiml, and R. Lackner, Real-scale CFD simulations of fire in single- and double-track railway tunnels of arched and rectangular shape under different ventilation conditions. Engineering Structures, 2014. 77: p. 193-206.

[11] Migoya, E., et al., Determination of the heat release rate inside operational road tunnels by comparison with CFD calculations. Tunnelling and Underground Space Technology, 2011. 26(1): p. 211-222.

[12] Ang, C.D., et al., Simulating longitudinal ventilation flows in long tunnels: Comparison of full CFD and multi-scale modelling approaches in FDS6. Tunnelling and Underground Space Technology, 2016. 52: p. 119-126.

[13] Chow, W.K., On smoke control for tunnels by longitudinal ventilation. Tunnelling and Underground Space Technology, 1998. 13(3): p. 271-275.

[14] Bubbico, R., B. Mazzarotta, and N. Verdone, CFD analysis of the dispersion of toxic materials in road tunnels. Journal of Loss Prevention in the Process Industries, 2014. 28: p. 47-59.

[15] Fluent, A., Academic Research. release 18.0.