*35th European Modeling & Simulation Symposium*
20th International Multidisciplinary Modeling & Simulation Multiconference

2724-0029 © 2023 The Authors.
doi: 10.46354/i3m.2023.emss.006

# Modeling of road tunnel airflows by FDS using tunnel model with a system of geometrical objects

Peter Weisenpacher[1,*]

[1]Institute of Informatics, Slovak Academy of Sciences, Dubravska cesta 9, 845 07 Bratislava, Slovakia

[*]Corresponding author. Email address: upsyweis@savba.sk

## Abstract

Computer simulation of airflow created by longitudinal ventilation in a highway tunnel can significantly contribute to design and computer simulation of tunnel fire scenarios and enable the evaluation of fire prevention measures. This paper examines the possibility of tunnel airflow modeling by Fire Dynamics Simulator (FDS). The influence of the tunnel geometry on airflow is modelled using cuboids regularly spaced in the tunnel tube along the tunnel centerline. The results of 5 tested models are compared with measurements conducted in the medium length tunnel Polana (Slovakia) as well as with the previous model of the tunnel based on the increased roughness of its walls. The measurements were conducted using a grid of 5 anemometers. The results of the simulation show a good agreement of steady-state bulk velocity values with the experimental values as well as slight improvement compared to the previous model. A slight improvement in the steady-state velocity values measured at the positions of the individual anemometers of the grid was also indicated.

**Keywords:** Tunnel ventilation system; airflow modelling; longitudinal ventilation; FDS

## 1. Introduction

Highway tunnels are an essential part of the road infrastructure, especially in mountainous regions. Although large-scale fires in tunnels are not frequent, their scale and the damage they cause to human life and property can be enormous, as can their impact on transport infrastructure. Therefore, great attention is paid to countermeasures in tunnels. One of the most common solutions, especially in medium-length tunnels, is longitudinal ventilation provided by jet fans, which induce air movement in one direction. A common strategy in single-tube double track tunnels is the effort to achieve the so-called smoke stratification (Klote and Milke 1992). If smoke remains stratified below the ceiling and the smoke layer does not reach the level of human head, tenable conditions for human life can be maintained and safe evacuation is possible. It is assumed that the conditions for smoke stratification can be maintained by reaching a specific target airflow velocity. Precise regulation of the flow velocity in the tunnel is therefore an essential part of fire safety measures.

CFD (Computational Fluid Dynamics) methods enabling computer simulation of various fire scenarios have become a common part of fire safety measures in structures. FDS (Fire Dynamics Simulator) (McGrattan, Hostikka, McDermott, Floyd, Weinschenk and Overholt, 2017) is a widely used code for different fire safety applications, such as airflows and smoke movement simulations (Valasek, 2013; Weng, Yu, Liu and Nielsen, 2014), evacuation (Glasa and Valasek, 2014) and design of smoke handling systems in buildings and various compartments.

Despite its computational efficiency, there is a relatively limited number of tunnel validation studies for FDS tunnel simulations. It may reflect the lack of trust in the possibilities of FDS to simulate specific features of tunnel geometry, especially tunnel curving and vaulted ceiling within the rectilinear calculation mesh used in FDS.

A special area of research is the simulation of airflows in the tunnel created by longitudinal ventilation in the case without fire. The correct simulation of these airflows and its validation is necessary condition of realistic simulation of tunnel fires. In the literature, there is still a lack of validation studies testing tunnel airflow models. This is mainly due to the difficulty and rarity of airflow measurements in tunnels, since such measurements would require the shutdown of the tunnel and associated financial and social costs.

The aim of this study is to create a model of the real 898 m long Polana tunnel using FDS, to explore the possibilities of its representation within the rectangular grid of FDS and to use it for the simulation of the airflow created by emergency ventilation in a tunnel without fire. The influence of the complex tunnel geometry on airflow is modelled using cuboids regularly distributed along the tunnel tube. Such model can be used for research and testing of fire scenarios by FDS. The results of the model are compared with the results of ventilation tests in the Polana tunnel as well as with the results of a previous model.

The article is structured as follows. Section 2 describes the state of the art in the field of computer simulation of tunnel fires and tunnel airflows. Section 3 describes the Polana tunnel in Slovakia and the ventilation tests in 2017. The model of the Polana tunnel is presented and its parameters are determined. In Section 4, the accuracy of model 2 is discussed. Finally, conclusions are drawn in Section 5.

## 2. State of the art

The first validation study related to the FDS simulation of tunnel ventilation used a full-scale fire experiment conducted as part of the Massachusetts Highway Department Memorial Tunnel Fire Ventilation Test Program (McGrattan, Hostikka, McDermott, Floyd, Weinschenk and Overholt, 2016). The test consisted of a single-point supply of fresh air through a 28 $m^2$ opening in a 135 m tunnel.

In (Alvarez-Coedo, Ayala, Cantizano and Wegrzynski, 2022), a coupled hybrid (1D/3D-CFD) modeling methodology using the FDS 6 (version 6.7.5) was validated by full-scale fire tests in the 1600 m long Runehamar tunnel. For 6, 66 and 119 MW fires, temperature profiles, centerline velocity, backlayering lengths and maximum temperatures upstream and downstream from the fire source were investigated.

In (Weng, Yu, Liu and Nielsen, 2014), a series of full-scale experiments with 1.35, 3 and 3.8 MW fires conducted in a short metro tunnel with mechanical ventilation system was described and then simulated by FDS (version 5.5). The smoke temperature and decay rate of the temperature distribution under the tunnel ceiling were investigated.

However, papers related specifically to the validation of an airflow generated by emergency ventilation in a real tunnel without fire using FDS are rare. Nevertheless, the accurate reproduction of tunnel airflow is crucial for proper modeling of the tunnel ventilation in the case of fire (Krol and Krol, 2021; Wang, Wang, Carvel and Wang, 2007). To increase the confidence in the ability of FDS to capture tunnel airflow, the average airflow velocity and velocity profile in the Dartford Tunnel (UK) were simulated using FDS 6 and compared with in situ measurements (Ang, Rein, Peiro and Harrison, 2016). The tunnel is 1430 m long with a circular cross-section with a diameter of 8.5 m. Although it was modeled as a tunnel with a square cross-section, the results correlate well with the measurements.

Realistic modelling of the tunnel requires a proper representation of curved concrete tunnel walls, S-shape tunnel geometry as well as objects which are not directly modelled, as their dimensions are too small in comparison with the mesh resolution (e.g., measuring devices, cameras, lighting and their supporting structures, niches). FDS limitation to rectilinear mesh does not allow modelling these features directly. (Weisenpacher and Valasek, 2021) solved this problem by a proper setting of ROUGHNESS parameter specifying velocity boundary conditions at rough solid surfaces using log law (Pope, 2000). The value of ROUGHNESS is determined to 70 mm for the Polana tunnel, higher than typical values of concrete roughness to capture the retarding effect of tunnel geometry on the airflow. (Glasa, Valasek, Weisenpacher and Kubisova, 2023) further contributed to the development of the model.

However, there is still a lack of validation studies investigating the bulk velocities in the tunnel created by longitudinal ventilation as well as the flow velocity profiles. Models based on other assumptions may bring more accurate modeling of the investigated phenomena.

Therefore, this study tests an alternative approach to tunnel modeling. Instead of increased roughness, the effect of tunnel geometry on airflow is modeled using the system of regularly placed cuboid objects under the tunnel ceiling. The values of the steady-state velocities for 3 ventilation modes are then compared with the measurement results in the Polana tunnel as well as with the results of simulations in (Weisenpacher and Valasek, 2021). Comparison of these two models can bring new findings and contribute to the accuracy of airflows modeling in the tunnel.

## 3. Materials and Methods

### 3.1. Fire Dynamics Simulator

Fire Dynamics Simulator (FDS) is a CFD-based simulation system for modelling fire and fire-driven fluid flows developed by the National Institute of Standards and Technology (NIST), USA in cooperation with the VTT Technical Research Centre of Finland. FDS numerically solves a form of conservation equations for low-speed thermally-driven flows with an emphasis on the smoke and heat transport from fire. The core algorithm is a very efficient explicit predictor-corrector scheme, second order accurate in space and time. FDS also includes models of fire related processes such as turbulence, thermal radiation, pyrolysis, combustion of the pyrolysis products, conductive heat transfer, etc.

Description of simulation scenario include the rectangular geometry of the scenario, i.e. dimensions of compartments, locations and dimensions of all relevant objects as well as material properties of object surfaces (ignition temperature, thermal conductivity, specific heat, density, emissivity, heat of evaporation, heat release rate, etc.) and other parameters.

FDS supports several models of parallelization of calculation The Message Passing Interface (MPI) model was used for this study due to its efficiency enabling to deal with significant computational requirements of a large compartment simulation.

The simulations were carried out on the SIVVP HPC cluster at the Institute of Informatics, Slovak Academy of Sciences, Bratislava (Slovakia). It is an IBM dx360 M3 cluster consisting of 54 computational nodes (23 Intel E5645 @ 2.4 GHz CPU, 48 GB RAM); the total number of cores is 648. The nodes are connected by the Infiniband interconnection network with the bandwidth of 40 Gbit/s per link and direction.

## 3.2. Ventilation tests in the Polana tunnel

The Polana tunnel (Pospisil and Ockajak, 2016; Danisovic, Sramek, Hodon and Hudik, 2017) is a single tube 898 m long bi-directional highway tunnel in Slovakia. It has a horseshoe cross-section of 10.8 m width and 6.8 m height; the area of cross section is 60.3 m². Two emergency lay-bys are located at 373 m (left side) and 635.6 m (right side) from the west tunnel portal, respectively. The lay-by niches are 50 m long and 2.2 m wide with the maximal height of vaulted ceiling of 7.8 m. Such tunnel dimensions are frequently used in road tunnels in Slovakia (Danisovic, Sramek, Hodon and Hudik, 2017). The tunnel has 2% ascending slope and the typical S-shape of the tunnel tube. It is equipped with various safety installations and measuring devices. An important part of the tunnel geometry is a supporting structure bearing lighting and other installations, located below the tunnel ceiling along the tunnel centerline.

The Polana tunnel is equipped with longitudinal ventilation consisting of four pairs of axial jet fans with 0.8 m fan wheel diameter and 3.7 m length (Figure 1). The pairs of jet fans are located at 101, 201, 716 and 801 m, respectively, at 5.1m height above the road. The jet fans are able to achieve the maximal volume flow of 19 $m^3.s^{-1}$ with 38.5 $m.s^{-1}$ airflow velocity at fan exit, exerting 850 N thrust in the main direction.

According to the Slovak technical regulation, the ventilation system is supposed to achieve longitudinal airflow velocity of 1.0–1.5 $m.s^{-1}$ within 120 s and to maintain it to maintain smoke stratification in case of fire, enabling the safe evacuation of people. The ventilation aim is to be achieved by stepless continuous airflow regulation by jet fans controlled by frequency converters. Airflow velocity used for the regulation is measured by three tunnel anemometers located at 340, 465 and 565 m, respectively.

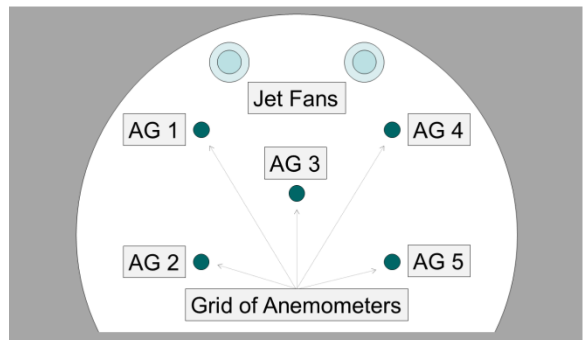


**Figure 1.** Cross-section of the Polana tunnel with jet fans and grid anemometers.

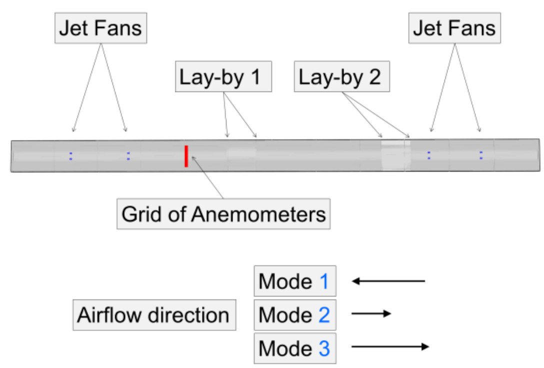


**Figure 2.** Scheme of the Polana tunnel.

In 2017, the ventilation tests in the Polana tunnel were conducted (Pospisil and Ockajak, 2017). The airflow velocity in the empty tunnel was measured by a grid of 5 anemometers (AG 1–5) located 300 m from the west (left) tunnel portal (Figures 1 and 2) and determined as an average of the measured grid anemometer velocities. Three modes of the ventilation operation were tested during 2 hours of experiment:

1. 8 jet fans operated at 100% power (westward direction)
2. 3 jet fans operated at 100% power, 5 jet fans operated at 30% power (eastward direction)
3. 8 jet fans operated at 100% power (eastward direction).

The aim of the measurements was to determine 3 steady-state airflow velocities corresponding to particular modes and use them for the determination of calibration factors for three anemometers installed in the tunnel.

### 3.3. Modelling of tunnel

The geometry of the Polana tunnel and its important components (the tunnel tube, two emergency lay-bys, jet fans, vertical traffic signs) for the FDS environment was modelled in detail, in a similar way as in (Weisenpacher and Valasek, 2021) (see Figure 2). The dimensions of the computational domain were set to 900 × 18 × 8.1 m in order to include the entire tunnel. As the sensitivity study in (Weisenpacher and Valasek, 2021) proved that 30 cm mesh resolution is optimal from the point of view of accuracy and efficiency of the computation, the resolution was used for all simulations.

A significant part of the objects in the Polana tunnel is made up of supporting structures carrying lighting, jet fans and other devices. The structures are placed above the centerline of the tunnel under its ceiling. This study examines the hypothesis that the effect of tunnel geometry on airflow occurs and it is caused primarily due to the interaction of the airflow with the supporting structures under the tunnel ceiling. As it is not possible to represent the supporting structures within the given 30 cm mesh resolution, their influence on airflows is modelled by a system of regularly placed cuboid objects under the tunnel ceiling (see Figure 3).

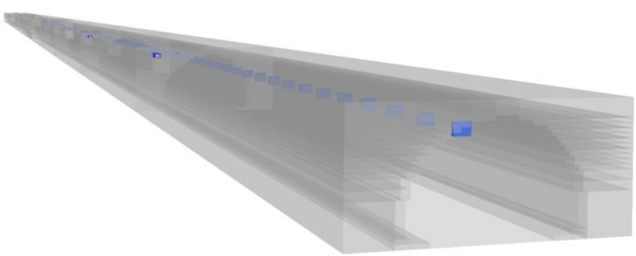

**Figure 3.** Cuboids under the tunnel ceiling, a view from the west tunnel portal.

Dimensions of the cuboids are 1 × 4 × 3 computation cells, i.e. 0.3 × 1.2 × 0.9 m. The dimensions were chosen to reflect approximately the dimensions of the structures in the tunnel they represent (see Figure 4). They are located at the height of 5.4 m.

**Figure 4.** Structures under the ceiling in the Polana tunnel.

The cuboids are placed in two different ways (see Figure 5). In the first half of the tunnel, the cuboids are placed above the tunnel centerline, shifted two cells to the right, with regular distances from each other. In the second half of the tunnel, the cuboids are placed above the tunnel centerline and shifted two cells to the left. Such arrangement reflects the curvature of the tunnel walls; cuboids are shifted to the convex wall, where a lower velocity is expected.

The cuboids are modeled within the FDS geometry using the MULT feature as follows:

1. the first half of the tunnel:

*&OBST XB=5.0,5.3,-1.2,0.0,5.4,6.3, ..., MULT_ID='m1'/*

*&MULT ID='m1', DX=5.4, I_LOWER=0, I_UPPER=80 /*

2. the second half of the tunnel:

*&OBST XB=455.0,455.3,0.0,1.2,5.4,6.3,... , MULT_ID='m2'/*

*&MULT ID='m2', DX=5.4, I_LOWER=0, I_UPPER=80 /*

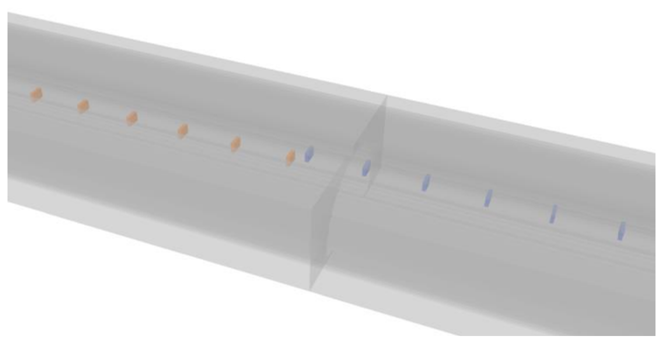

**Figure 5.** Two types of cuboids locations in the central sector of the tunnel.

The description includes the coordinates of the first cuboid (at 5 and 455 meters of the tunnel for the first and second halves of the tunnel, respectively), the distance between two consecutive cuboids (5.4 m) and their number (from 0 to 80, i.e. 81 cuboids in total). According to such settings, the distance between the last cuboid of the first half of the tunnel and the first cuboid of the second half of the tunnel may be significantly different than 5.4 m (see Figure 5). Note that the values of 5.4 m and the number of cuboids 81 are used only as an example here; their exact number needs to be determined later.

**Table 1.** Simulations settings.

| | |
|---|---|
| Ambient temperature | 6°C |
| Mesh resolution | 30 cm |
| Computational domain | 900 × 18 × 8.1 m |
| Mesh decomposition | 12 × 1 × 1 (12 identical meshes) |
| Cells | 3000 × 60 × 27 (4,860,000 total) |
| Turbulence | LES, Deardorff model |
| Boundary conditions | CONCRETE for tunnel walls and road (ROUGHNESS 0.001 m) OPEN for tunnel portals (DYNAMIC_PRESSURE -6.3 Pa to -1 Pa) |
| Time step | Variable, typically 0.006 s for active jet fans, CFL 0.08 |
| Simulation time | 7180 s |
| Simulation elapsed time | 443 to 461 hours |

The jet fans operation is modelled by the HVAC feature used for modelling of Heating, Ventilation, and Air Conditioning systems in FDS (McGrattan, Hostikka, McDermott, Floyd, Weinschenk and Overholt, 2017). The fan is modelled as a one-cell thick OBSTruction of 0.9 × 0.6 m dimensions with two VENTs associated with it, representing the jet fan intake and outlet with the volume flow prescribed by HVAC (McGrattan, Hostikka, McDermott, Floyd, Weinschenk and Overholt, 2017; Ang, Rein, Peiro and Harrison, 2016). The volume flow set in HVAC is adjusted to 19.75 $m^3.s^{-1}$, as a consequence of the different area of the modelled (0.54 $m^2$) and real (0.5 $m^2$) jet fan (see discussion in (Weisenpacher and Valasek, 2021)). The shroud of every jet fan is created by four one mesh cell thick OBSTructions of 3.9 m length. The value of volume flow in HVAC corresponds to the maximal performance of particular jet fan. The time dependence of the volume flow rate of jet fans in operation is set by the RAMP feature according to their performance during the ventilation tests.

Simulations of 7180 s of the ventilation tests were performed for all tested models. The 3 time intervals of stabilized jet fan performance, 755–1255 s, 2515–3130 s and 5165–5760 s were used for 3 ventilation modes, respectively. Main settings of FDS simulations are summarized in Table 1. Note that the dynamic pressure values on the portals of the tunnel were set in accordance with (Weisenpacher and Valasek, 2021) to represent natural airflows. Dynamic pressure induces variable non-zero airflows in the tunnel also in the intervals between ventilation modes as was observed during the tests.

### 3.4. Definition and evaluation of models

Five models varying in the number and density of cuboids were tested and evaluated. Their parameters are summarized in Table 2. The steady-state velocities in all 3 operating modes (3 values) as well as the steady-state velocities at the locations of individual grid anemometers (3×5 values) were used to evaluate the accuracy of individual models.

**Table 2.** Settings of simulation scenarios.

| | Cuboids | Cuboids (left/right) | Mutual distance [m] |
|---|---|---|---|
| Model 160 | 160 | 80/80 | 5.4 |
| Model 166 | 166 | 83/83 | 5.4 |
| Model 170 | 170 | 85/85 | 5.1 |
| Model 174 | 174 | 87/87 | 5.1 |
| Model 178 | 178 | 89/89 | 5.1 |

Airflow velocity in the simulation was determined as a mean of velocities at the positions of grid anemometers, consistent with experiment as well as with (Weisenpacher and Valasek, 2021). Time intervals 920–1255 s, 2750–3130 s and 5350–5750 s corresponding to steady-state flow were used to determine steady-state velocities as a mean of all values in the interval, recorded with 5 s frequency.

A sum of squares of the deviations of the 3 simulated steady-state airflow velocities from the corresponding velocities obtained experimentally (denoted by $D_3$) was determined for each model, as well as sum of squares of the deviations of the 15 simulated steady-state velocities for AG 1–5 positions from the experimental steady-state velocities (denoted by $D_{15}$). Both values were used for evaluation of simulation accuracy, while the best accuracy corresponds to minimal values of $D_3$ and $D_{15}$. They were also compared with corresponding values for model of (Weisenpacher and Valasek, 2021), denoted by Model 0.

## 4. Results and Discussion

Table 3 contains experimentally determined steady-state velocities for 3 ventilation modes and the corresponding values of steady-state velocities determined by individual models. The table also shows the values determined by Model 0, which enables the comparison of both presented approaches to tunnel modeling.

Optimization procedure started with Model 160, containing 160 cuboids. Slightly higher velocities achieved by this model than the ones measured during ventilation tests, as well as high values of $D_3$ and $D_{15}$ led to the necessity for increasing the number of cuboids in subsequent models settings. The fifth model (Model 178) with 178 cuboids with 5.1 m mutual distances finally achieved $D_3$ and $D_{15}$ values lower than the ones determined in Model 0.

It can be seen from Table 3 that the process of optimization was not straightforward. Increasing the number of cuboids may lead to a necessity of smaller distances between them, which negatively affects the model's accuracy (see $D_3$ values for the Model 166 and Model 170, in which mutual distance of cuboids decreased from 5.4 to 5.1 m). Another increase of cuboids number with the new mutual distance was necessary to achieve further accuracy improvement (see Model 170 and Model 174). In other words, the $D_3$ and $D_{15}$ dependences on cuboids number are not monotonous functions.

In Figure 6, airflow velocities for Model 178 are compared with the values obtained during the ventilation tests. For all 3 modes (see Figure 6), the differences between experimental and simulated values are very small.

**Table 3.** Experimental and simulated values of steady-state airflow velocities for three ventilation modes [m.s$^{-1}$], absolute [m.s$^{-1}$] and relative [%] differences between experimental and simulated values, $D_3$ [m$^2$.s$^{-2}$] and $D_{15}$ [m$^2$.s$^{-2}$].

| | Mode 1 | Mode 2 | Mode 3 | $D_3$ | $D_{15}$ |
|---|---|---|---|---|---|
| Experiment | -5.61 | 3.42 | 5.41 | | |
| Model 0 | -5.72 | 3.47 | 5.43 | 0.016 | 5.27 |
| | -0.11 | 0.05 | 0.02 | | |
| | 2.0% | 1.5% | 0.4% | | |
| Model 160 | -5.80 | 3.49 | 5.60 | 0.079 | 5.03 |
| | -0.19 | 0.07 | 0.19 | | |
| | 3.4% | 2.0% | 3.5% | | |
| Model 166 | -5.74 | 3.46 | 5.55 | 0.039 | 5.07 |
| | -0.13 | 0.04 | 0.14 | | |
| | 2.3% | 1.2% | 2.6% | | |
| Model 170 | -5.75 | 3.45 | 5.54 | 0.040 | 4.23 |
| | -0.14 | 0.03 | 0.13 | | |
| | 2.5% | 0.9% | 2.4% | | |
| Model 174 | -5.72 | 3.43 | 5.49 | 0.020 | 4.31 |
| | -0.11 | 0.01 | 0.08 | | |
| | 2.0% | 0.3% | 1.5% | | |
| Model 178 | -5.64 | 3.41 | 5.44 | 0.002 | 4.11 |
| | -0.03 | -0.01 | 0.03 | | |
| | 0.5% | -0.3% | 0.6% | | |

Although the optimization procedure determined the most accurate model among the 5 tested models, the accuracy of all 5 models is sufficient for practical needs. The absolute value of the absolute difference did not exceed the value of 0.19 m.s$^{-1}$ in any of them (Model 160, Mode 1 with relative difference of 3.6%), which is less than the accuracy of the used grid anemometers.

The most accurate of the used models, Model 178, achieved accuracy even below the limit of 0.04 m.s$^{-1}$ in all 3 modes, which is a value far lower than practical needs. It proves the ability of the model to determine the bulk airflow velocities. From a purely mathematical point of view, this value is better than the corresponding value in Model 0 (0.12 m.s$^{-1}$); however, from the point of view of practical needs, it is more correct to consider both models to be comparably accurate in this regard.

The relative differences of steady-state velocities on the particular anemometers AG 1–5 in Model 0 are higher than those for steady-state bulk velocities (see discussion in (Weisenpacher and Valasek, 2021)). It can also be seen from the value of $D_{15}$, which is an order of magnitude larger than $D_3$. The same is true for the models examined in this paper (see Table 3). However, the approach tested here may have the potential to partially reduce these inaccuracies, as can be seen from the $D_{15}$ value, which improved from 5.27 for Model 0 to 4.11 for Model 178. Although this improvement is not considerable, it is not negligible. Modelling the velocity profile requires further research and analysis. It can be expected that a different and potentially more appropriate distribution of cuboids could lead to further refinement of the model.

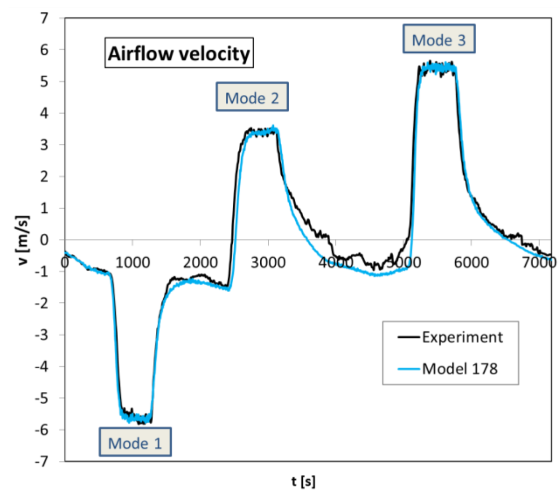


**Figure 6.** Airflow velocity during ventilation tests and by Model 178.

## 5. Conclusions

This paper tested the possibility of FDS modeling the airflows in the Polana tunnel created by longitudinal ventilation, whereby the effect of the tunnel geometry on the flow was modeled using a set of cuboids located under the tunnel ceiling along its centerline. The results were compared against the experimental measurement of the airflow velocities using a grid of 5 anemometers for 3 investigated ventilation modes, as well as against the results of the model (Weisenpacher and Valasek, 2021). 5 models with different numbers of cuboids were examined, and the number 178 was found to be the most suitable.

The model achieves significant accuracy in determining steady-state bulk airflow velocities, with differences lower than the measurement accuracy of used grid anemometers. All three relative differences for particular ventilation modes are below 1%. Such accuracy is slightly better than the accuracy achieved in (Weisenpacher and Valasek, 2021) and sufficient for practical needs. A slight improvement which needs to be analyzed further was also indicated in the values of steady-state velocities at the positions of individual grid anemometers.

Further research should focus on validation of the velocity profile of the given model, especially in the upper part of the tunnel where it is significantly influenced by cuboids. However, certain limitations of accuracy in this area are inevitable, as it is not possible to accurately model the influence of sub-grid scale objects.

The results of this study indicate the possibilities of simulating the geometry of the tunnel and its influence on the airflow using a system of geometrical objects. Investigating their appropriate dimensions and distribution could lead to the further refinement of the model, including simulated velocity profile, as well as to the improvement of the possibility of simulating fire scenarios for practical needs.

## Funding

This work was supported by the Slovak Scientific Grant Agency VEGA under the contract 2/0135/23 "Intelligent sensor systems and data processing".

## Acknowledgements

The author would like to thank Peter Schmidt (National Motorway Company, Slovakia) for information about technical specifications of road tunnels and Petr Pospisil (IP Engineering GmbH, Switzerland) for the measurement data.

## References

Alvarez-Coedo, D., Ayala, P., Cantizano, A. and Wegrzynski, W. (2022). A coupled hybrid numerical study of tunnel longitudinal ventilation under fire conditions. *Case Stud. Therm. Eng.*, 36:102202.

Ang, C.D., Rein, G., Peiro, J. and Harrison, R. (2016). Simulating longitudinal ventilation flows in long tunnels: Comparison of full CFD and multi-scale modelling approaches in FDS6. *Tunn. Undergr. Space Technol.*, 52:119–126.

Danisovic P., Sramek J., Hodon M. and Hudik M. (2017). Testing measurements of airflow velocity in road tunnels. *MATEC Web of Conferences*, 117:00035.

Glasa, J. and Valasek, L. (2014). Study on applicability of FDS+Evac for evacuation modeling in case of road tunnel fire. *Res. J. Appl. Sci. Eng. Technol.*, 7:3603-3615.

Glasa, J., Valasek, L., Weisenpacher, P. and Kubisova, T. (2023). Improvement of modeling velocity of airflow created by emergency ventilation in a road tunnel using FDS 6. *Appl. Sci.*, 13:2762.

McGrattan, K., Hostikka, S., McDermott, R., Floyd, J., Weinschenk, C. and Overholt, K. (2016). Fire Dynamics Simulator, Technical Reference Guide, Validation, 6th ed. NIST: Gaithersburg, MD, USA, Volume 3.

McGrattan, K., Hostikka, S., McDermott, R., Floyd, J., Weinschenk, C. and Overholt, K. (2017). Fire Dynamics Simulator, User's Guide, 6th ed. NIST: Gaithersburg, MD, USA.

Klote J. and Milke, J. (1992). Design of smoke management systems. American Society of Heating, Refrigerating and Air Conditioning Engineers. Atlanta, Georgia, USA.

Krol, A. and Krol, M. (2021). Some tips on numerical modelling of airflow and fires in road tunnels. *Energies*, 14:2366.

Pope, S.B. (2000). Turbulent flows. Cambridge University Press: Cambridge, UK.

Pospisil, P. and Ockajak, R. (2016). Polana Tunnel—Tunnel and escape tunnel ventilation. Technical Report, IP Engineering GmbH: Munchenstein, Switzerland (In Slovak).

Pospisil, P. and Ockajak, R. (2017). Ventilation Tests in Polana and Svrcinovec tunnels. Technical Report, IP Engineering GmbH: Munchenstein, Switzerland, pp. 154–196 (In Slovak).

Valasek, L. (2013). The use of PyroSim graphical user interface for FDS simulation of a cinema fire. *Int. J. Math. Comput. Simul.*, 7:258-266.

Wang, F., Wang, M., Carvel, R. and Wang, Y. (2007). Numerical study on fire smoke movement and control in curved road tunnels. *Tunn. Undergr. Space Technol.*, 67:1–7.

Weisenpacher, P. and Valasek, L. (2021). Computer simulation of airflows generated by jet fans in real road tunnel by parallel version of FDS 6. *Int. J. Vent.*, 20:20–33.

Weng, M.C., Yu, L.X., Liu, F. and Nielsen, P.V. (2014). Full-scale experiment and CFD simulation on smoke movement and smoke control in a metro tunnel with one opening portal. *Tunn. Undergr. Space Technol.*, 42:96–104.