Simulation-Aided Development of a Compact Local Ventilation Unit with the Use of CFD Analysis

Petr Zelenský [a,b], *Martin* Barták [a,b], *Vladimír* Zmrhal [a,b], *Juraj* Mázik [c]

[a] Department of Environmental Engineering, Faculty of Mechanical Engineering, Czech Technical University in Prague, Czech Republic, petr.zelensky@cvut.cz

[b] University Centre for Energy Efficient Buildings, Czech Technical University in Prague, Czech Republic.

[c] RECUAIR, Kralupy nad Vltavou, Czech Republic

Abstract. The current emphasis on the renovation of existing buildings to meet EU energy efficiency targets brings, in addition to energy savings and related CO2 reduction, also some negative issues. One of them can be the lack of fresh air supply caused by increased air tightness of the building envelope after its insulation and renovation or change of windows. The easy solutions are decentralized units for local ventilation, which can be installed during fast renovations in selected rooms without major building modifications. Controlled ventilation then ensures the delivery of a sufficient amount of fresh air to meet current standards and, at the same time, creates a healthy and comfortable environment for occupants. The paper demonstrates the practical use of CFD simulations for the development of a new type of compact small ventilation unit for local ventilation of rooms with heat and humidity recovery. An increase in the device efficiency and a reduction in acoustic power, while maintaining its very compact dimensions, were achieved with the help of the numerical study. The paper shows the possibility of using CFD analysis during the development of new HVAC appliances. It describes the preparation of the numerical model of the device, presents the simulation approach, including the calculation settings, and discusses device optimization based on variant numerical analyses in ANSYS Fluent. The initial prototype design of the unit was optimized following the findings from the numerical analysis, and it was verified by CFD study that the proposed adjustments were appropriate and that the expected results were achieved. In a separate CFD study, the use of different types of diffusers at the air outlet from the supply duct to the room was addressed. It was recommended to use adjustable nozzles, which allow one to direct the air flow into the room according to the user's preference. Consequently, it was verified that the ventilation unit meets the hygienic noise limits, both for day operation and for night operation with reduced power.

Keywords. Local ventilation, compact ventilation unit, development, optimization, CFD, numerical analysis.
DOI: https://doi.org/10.34641/clima.2022.194

1. Introduction

The lack of fresh air supply caused by the increase in the air tightness of the building envelopes after their renovation is a major concern of HVAC engineering today. Additional insulation of walls and renovation or change of windows of existing building stocks is driven by the EU energy efficiency targets that lead to energy savings and related CO_2 reduction. However, in some cases of fast renovations that target only the building envelope, proper ventilation and fresh air delivery can be ignored. This leads to inadequate indoor air quality, humidity problems, and other related issues. The easy solutions to improve indoor environment quality are decentralized local ventilation units, which can be installed during such fast renovations, without major building modifications. Controlled ventilation then ensures the delivery of a sufficient amount of fresh air to meet current standards and, at the same time, creates a healthy and comfortable environment for the occupants.

The paper demonstrates the practical use of computational fluid dynamics (CFD) simulations for the development of the compact ventilation unit for local ventilation of rooms. It is a decentralised unit without the need for condensation drainage and any necessary connections to other building systems. The visible part of the unit (Fig. 1, top) is located on the inner wall of the room and contains supply and exhaust fans, air filters, and control system of the unit. The cylindrical part on the back side of the unit (Fig. 1, bottom) passes through the external wall of the building and ends on the facade.

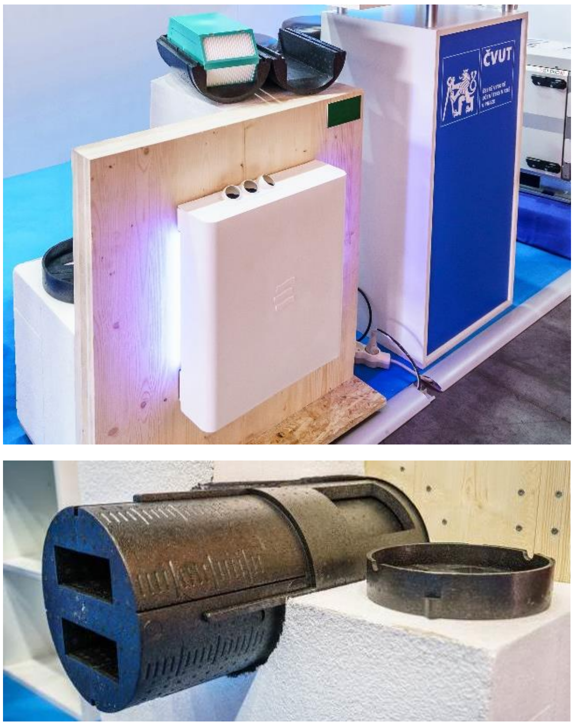

Fig. 1 - Compact ventilation unit [1]

The cylindrical part of the unit houses a rotary recuperation heat exchanger driven by an actuator, which turns it 180° at intervals determined according to the outdoor air temperature. This way, an alternation of the flow of supply outdoor air and exhaust degraded air is ensured, which enables humidity recovery throughout the year and prevents icing of the exchanger in the winter season. High efficiency of heat and moisture recovery is achieved all year round.

A numerical CFD study of airflow in a complex system of supply and exhaust air channels of the ventilation unit was performed. CFD simulations provide detailed information about pressure, velocity, turbulence, and other quantities on the network of computational cells (control volumes) that divide the area of interest. The motivation for using this method was to reduce the cost of the development, manufacturing, and operation of the device. Effective optimization of the complex geometry of supply and exhaust air channels of the ventilation unit would be very challenging without the use of CFD simulations, which provide otherwise inaccessible information and relatively easy possibility to analyse and compare variant design solutions.

The paper describes the preparation of a numerical model of the device for CFD simulation, presents the simulation approach, including the calculation settings, and discusses the device optimization based on variant numerical analyses in ANSYS Fluent. The methods of simulation of the fan impeller rotation and approximation of the filtration material in the model are discussed in detail. The results of the numerical analysis served as feedback to the design team during the development and optimization of the prototype device. It provided the basis for changes in the internal arrangement of the unit, the purpose of which was to minimize the occurrence of local pressure losses and possible sources of aerodynamic noise. It helped the design team to reach a design solution that brings higher energy efficiency and lower acoustic power generated by the device, taking into account the compact dimensions of the unit.

The numerical analysis described in this paper was followed by a measurement of the noise and volume flow rate of the unit under laboratory conditions. The study was part of a project supported by the EU under the Operational Program Enterprise and Innovation for Competitiveness.

2. Numerical model of the unit

The numerical model of the examined device was created on the basis of the 3D prototype geometry provided by the design team; see Fig. 2. The dimensions of the device were 455 × 425 × 105 mm, the cylindrical part had diameter of 260 mm.

The variant numerical analysis was solved iteratively, following these seven steps:

1. import of the CAD design geometry into the ANSYS SpaceClaim software;
2. adjustment of the geometry for CFD simulations (software: SpaceClaim);
3. preparation of the numerical mesh (software: ANSYS Fluent Meshing);
4. CFD simulations of airflow (software: ANSYS Fluent);
5. analysis and discussion of the results;
6. proposal of device design optimization;
7. CFD analysis of the optimized geometry (repeating steps 1 to 6).

Fig. 2 - Prototype geometry of the ventilation unit

2.1 Preparation of the model for CFD

The prototype geometry was available in the universal format STEP and could be imported directly into the ANSYS SpaceClaim as a spatial model. The initial geometry was adjusted for the CFD simulations. After quality control of the obtained spatial model and correction of minor deficiencies (especially colliding bodies), the simplifications listed below were made. It was expected that the simplifications and geometry adjustments should have minimal influence on the airflow in the ventilation unit. The modified geometry is shown in Fig. 3.

- All the bolts were removed; the empty holes for the removed parts were filled;
- technological fillets and cutouts of sheet steel parts of the device were removed;
- the cylindrical part of the unit housing the heat exchanger was removed (heat exchanger was approximated in the simulation by a boundary condition);
- overlapping surfaces were corrected;
- small surfaces of the air channel walls were removed;
- the curved surfaces of the air channel walls were simplified.

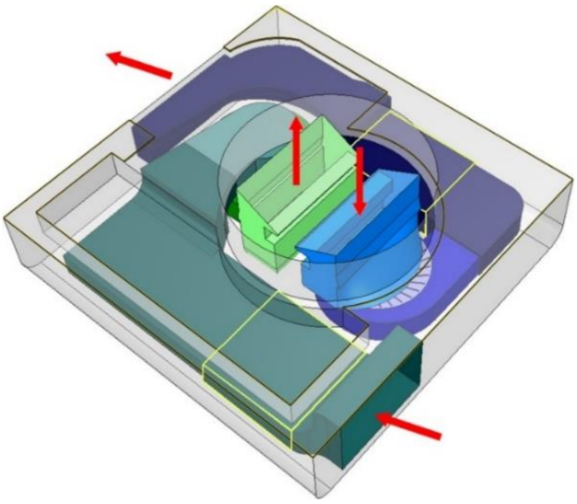

Fig. 3 - Ventilation unit model as adapted for CFD; supply channel (in blue), exhaust channel (in green); domains with filters marked by yellow line; airflow direction marked by red arrows

The simulation study analysed the airflow in the supply channel (see Fig. 3, in blue) and the exhaust channel (Fig. 3, in green). The geometry of the air channels is quite complex. The supply air is drawn on the façade and passes through the heat exchanger through the wall of the building. Then it flows through a small radial fan and is distributed to the room by a supply channel containing a filter. Exhaust air is drawn from the room on the opposite side of the ventilation unit. It passes through the exhaust channel with the filtration material and flows through the exhaust fan and heat exchanger, where it transfers its heat to the supply air, to the façade outlet. It is discharged to the outdoor environment.

The aim of the study was to predict the characteristics of the airflow in the ventilation unit, including the airflow through the rotating radial fans and the air filters. Both channels have a unique geometry to maintain as compact dimensions of the unit as possible and at the same time achieve sufficient noise attenuation (their walls are partially lined with noise-absorbing material, although with limited thickness). The CFD study indicates critical points of the geometry with risk of increased generation of aerodynamic noise, such as regions with local increase of velocity, points of air flow separation, etc.

2.2 Preparation of the numerical mesh

CFD simulations are usually based on the finite volume method, i.e., division of the investigated domain into a large number of consecutive control volumes and the solution of conservation equations of mass, momentum, and energy for each of them. The quality of the numerical mesh significantly affects the convergence of the simulation and the accuracy of the results. Therefore, attention must be paid to the meshing part of the model preparation.

The geometry of the supply and exhaust air channels was imported into the ANSYS Fluent Meshing. The volumetric model was automatically converted to a set of surfaces meshed by a triangular grid with predefined parameters. The surface mesh serves as a basis for the following discretization of the domain volume. Therefore, it must be sufficiently fine to capture all the important geometric details. It is also necessary to check its quality and, if necessary, repair problematic cells.

As the next step, a hybrid polyhexcore volume mesh with a prismatic boundary layer mesh on the selected surfaces was created. The mesh combines high-quality octree hexahedron in the bulk region, and isotropic polyhedral cells near the walls; see for example Fig. 4.

Each of the channels was meshed and simulated separately (airflows in the channels do not affect each other). The maximum size of mesh cells was globally set to 2 mm; at critical locations for simulation, the general cell size was manually corrected to provide a finer computational mesh (e.g., around radial fans, near the walls, etc.). The smallest cell size was 0.2 mm. A prismatic mesh region was created, with a height of 10 cells on the fan blades and a height of 6 cells on the surfaces of the fan housings and on the walls of the channels. The resulting mesh had $6{,}5 \cdot 10^6$ million computational cells for the case of the supply channel and $8{,}3 \cdot 10^6$ million cells for the case of the exhaust channel.

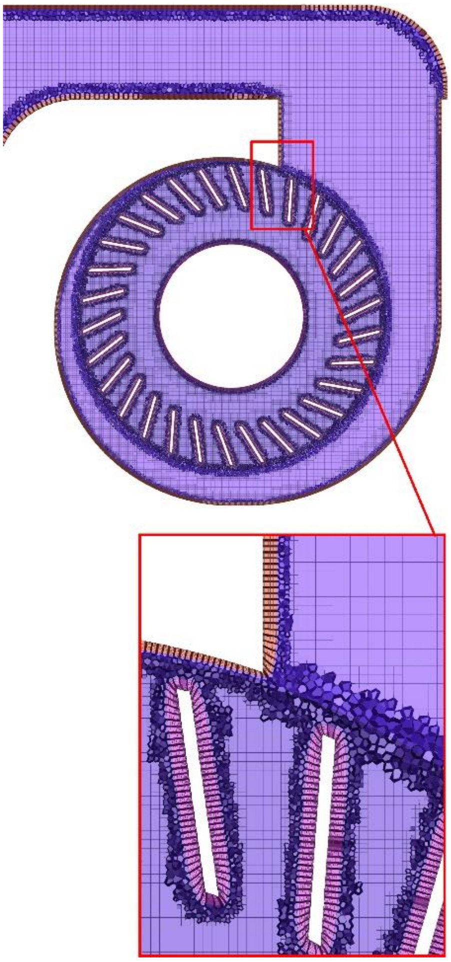

Fig. 4 - Computational mesh for CFD simulations

3. Numerical solution

The CFD simulations were solved using the software ANSYS Fluent 19.0. as a flow of incompressible air, without considering the energy equation. The two-equation model k-ε according to Shih et al. [2] (so called k-ε Realizable) was used for all calculations. It was previously shown to be optimal for simulation of similar types of devices with rotating fan impellers [3]. The flow in the near-wall regions was calculated by integration of the governing equations in the viscous sublayer, with the use of the Enhanced Wall Treatment method. The PRESTO! scheme was chosen for the discretization of the pressure equation as it is recommended for solving rotary machines [4]. The convective terms of the equation were solved using second order upwind scheme. The pressure and velocity fields were coupled by the SIMPLE algorithm.

The pressure loss of the filters was approximated in the simulation by porous layers added to the numerical models. The characteristics of the porous elements were established on the basis of documentation provided by the filter manufacturer. The PorZo software add-on [5], an ANSYS Customization Toolkit (ACT) extension of the Fluent software, was used to prescribe the characteristic pressure-volume curve of the filter to the porous layers.

The pressure loss of the heat exchanger was determined according to the manufacturer's documentation. The RecAir RS220 exchanger was used in the prototype of the device. The pressure drop was prescribed as boundary conditions at the inlet to the supply channel and at the outlet from the exhaust channel.

The boundary conditions facing the room were considered as open space, that is, the static pressure at the inlet to the exhaust channel and the outlet from the supply channel was 0 Pa, the intensity of turbulence was 0.1 % and the turbulent viscosity ratio 0.01.

The correct approximation of the fan impeller rotation is the fundamental part of the CFD simulation of the ventilation unit. There are two different methods available in ANSYS Fluent [4, 6, 7, 8], while the MRF method was used for the approximation of the fan impeller rotation in the presented study:

- **Multiple Reference Frame (MRF)** – airflow is calculated in one single moment of the rotation, i.e. in one single impeller position (the method can also be referred to as the "frozen rotor approach"). It does not account for the relative motion of a moving zone with respect to adjacent zones. The impeller rotation is approximated by prescribing the tangential components of the velocity to the specified volume rotating at a chosen angular velocity. The result of the simulation is a stationary image of the flow in the domain. The MRF method is not recommended to be used in unsteady simulations.
- **Sliding Mesh** – airflow is considered as unsteady. The rotation of the wheel is simulated directly, that is, the relative motion of a moving zone (fan impeller) with respect to adjacent zones is considered, and continuous change of the geometry occurs in the CFD simulation. The advantage of this method compared to the MRF approach is the potential higher accuracy of the results obtained. The disadvantage is considerably higher demand on computational time and difficult post-processing of the obtained results arising from the large number of data files obtained for individual positions of the fan impeller during its rotation.

4. CFD Simulation and evaluation of first Results

All simulations were calculated for one operation point of the ventilation unit with the required airflow of 45 m^3/h through both channels, which was the maximum design airflow of the prototype unit. The corresponding pressure loss of the heat exchanger was 112.5 Pa and the fan speed was 3,920 RPM. The numerical study was divided into two separate tasks: (i) simulation of the supply channel and (ii) simulation of the exhaust channel. The velocity fields were analysed for each of the channels.

Fig. 5 shows the velocity isosurfaces in the supply channel. The airflow velocity at the fan impellers reaches approximately 33 m/s. The velocity in the channel is in range from 1.5 to 12 m/s, while high velocities can be observed especially in the restricted part of the channel before the outlet to the room and near the 90° bend of the channel behind the fan. Moreover, the 90° bend has a sharp inner corner. Fig. 6 shows the disturbance of the airflow trajectories in the mentioned areas.

Another concern was the noise generated by the fan and its propagation through the supply channel to the room, as it was not possible to equip the unit with silencers. Noise attenuation was addressed by adding a layer of noise-absorbing material on the inner walls of the channels and by considerate design of the internal airways. The position of the fan in the prototype geometry was found to be not optimal and the issue was further addressed during the optimization of the ventilation unit.

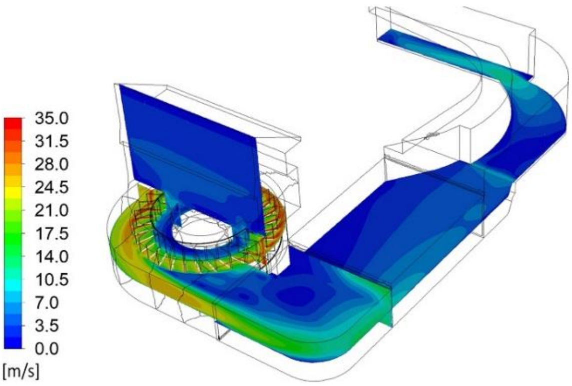

Fig. 5 – Velocity isosurfaces in the supply channel

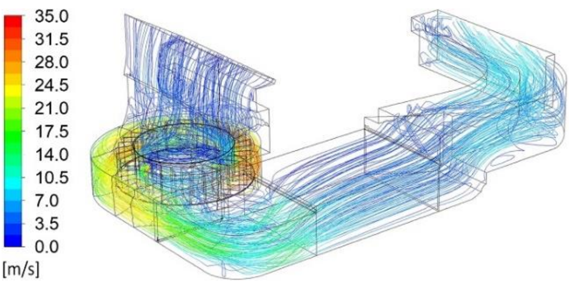

Fig. 6 - Airflow pathlines in the supply channel

The complex geometry of the supply channel near the outlet of the ventilation unit to the room was studied in detail. The narrow part of the channel and sharp bend before the outlet to the room lead to relatively high air flow velocity of up to 9 m/s; see Fig. 7. This brings a risk of high aerodynamic noise. Furthermore, the velocity profile of the airflow at the outlet opening is not uniform. With the given geometry of the supply channel, some areas show a very high outlet velocity, while only the minimum airflow is achieved in the remaining area of the outlet. Moreover, the bend of the channel before the outlet causes a high directivity of airflow into the room. These issues were faced during the optimization phase of the study.

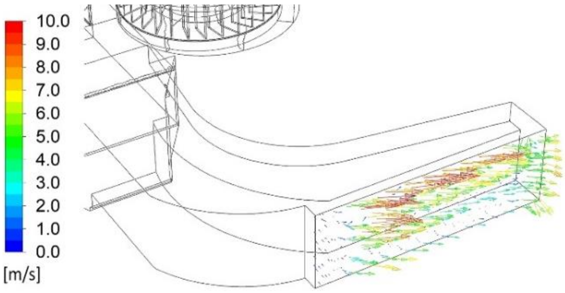

Fig. 7 - Detail of the velocity field at the outlet to the room

The geometry of the exhaust channel is less complex in comparison to the geometry of the supply channel. Also, most of the channel is on the suction side of the fan. Thus, it shows a more uniform airflow, see Fig. 8. Significant disturbance of airflow is apparent only at the section after the fan, before the inlet to the heat exchanger. It was recommended to round the sharp corners of several bends in this region. The velocity of airflow in the channel is in range from 3 to 5 m/s, the velocity at the fan reaches 33 m/s.

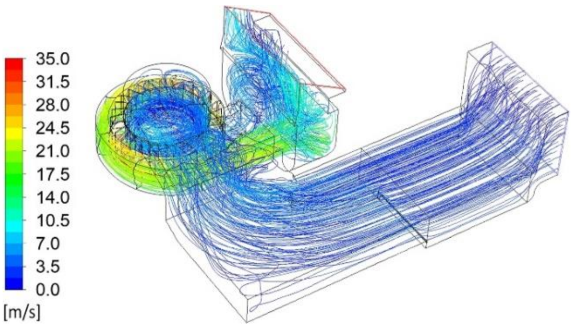

Fig. 8 - Airflow pathlines in the exhaust channel

5. Ventilation unit optimization

Critical points of the prototype geometry were identified on the basis of the initial CFD simulations and modification of the supply and exhaust channel geometry was proposed. The effect of the modifications on airflow was analysed by a subsequent numerical study.

Compared to the prototype geometry of the unit, the following optimizations were applied (see also Fig. 9):

1. geometry modification at the inlet from the room;
2. fillets on the sharp edges of the exhaust channel in front of the fan;
3. fillet on the sharp corner of the supply channel after the fan;
4. change of the supply channel fan orientation (the fan was rotated by 90° in order to reduce the noise emitted to the room);

5. adding an air diffuser at the outlet of the ventilation unit to the room; a separate variant study was performed to choose an optimal air diffuser, while four different configurations were compared; see Fig. 10:
   a) outlet without a diffuser (original solution);
   b) outlet with a perforated diffuser;
   c) outlet with directional vanes;
   d) outlet with adjustable nozzles.

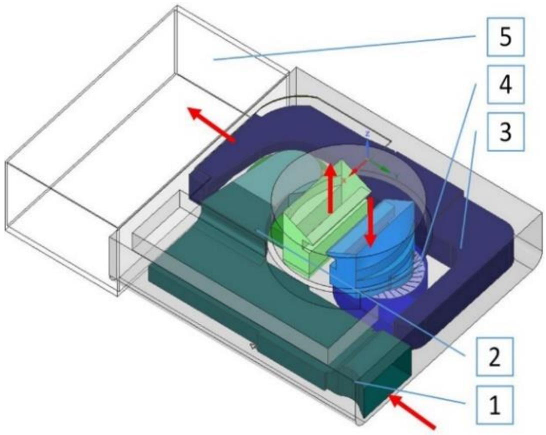

Fig. 9 - Optimized geometry of the ventilation unit

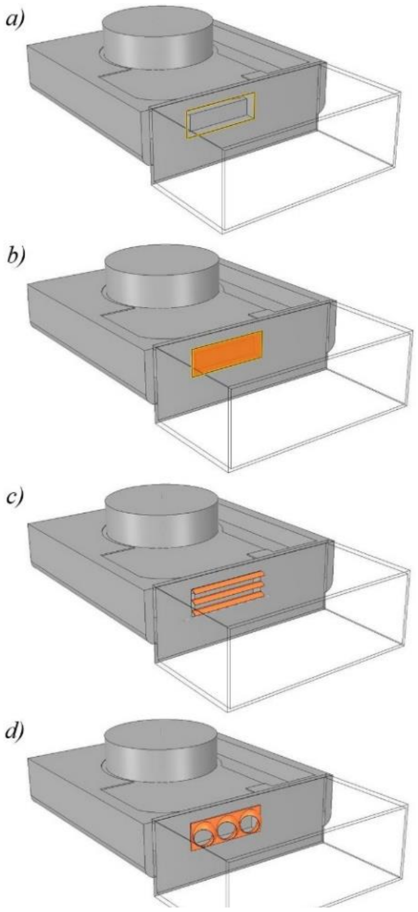

Fig. 10 - Variant study of different HVAC diffusers

The numerical study of the airflow after geometry optimization was carried out following the same methodology as the CFD simulations with the prototype geometry. The spatial model of the supply channel was extended by a part of the surrounding space, to study the directional characteristics of the airflow into the room. The numerical mesh was created with the same parameters. In the case of the supply channel, it had $11 \cdot 10^6$ computational cells and in the case of the exhaust channel $8.1 \cdot 10^6$ computational cells. The CFD simulations were set identically. Calculations were performed for one operating state of the unit, with volume air flow rate through the device of 45 m³/h and corresponding pressure loss of the heat exchanger of 112.5 Pa.

6.3 CFD analysis of the optimised design

By optimization of the supply channel geometry, a more uniform airflow was achieved, see Fig. 11 and Fig. 12. The velocity in the channel is in the range from 1.5 to 15 m/s, similar to the simulation with the prototype geometry. However, a high velocity can be observed mainly in the narrow part of the channel immediately behind the fan. This region is, due to the rotated position of the fan, shielded from the outlet to the room. The fan noise as well as the aerodynamic noise of the fast airflow are more effectively absorbed by the walls, which are lined by the sound insulation material.

However, even after the geometry optimization, the airflow at the outlet to the room was still not optimal, with very high directionality. This issue was solved in a separate variant study presented in the following chapter.

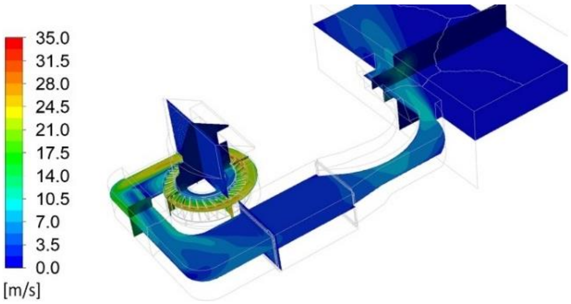

Fig. 11 - Velocity isosurfaces in the supply channel after optimization

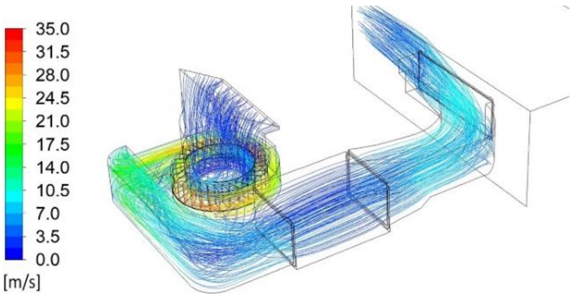

Fig. 12 - Airflow pathlines in the supply channel after optimization

In the case of the exhaust channel, the applied optimisations did not have a significant effect on the airflow, as the flow was already uniform in the previous case of the prototype geometry. The airflow pathlines after the geometry optimization shows Fig. 13.

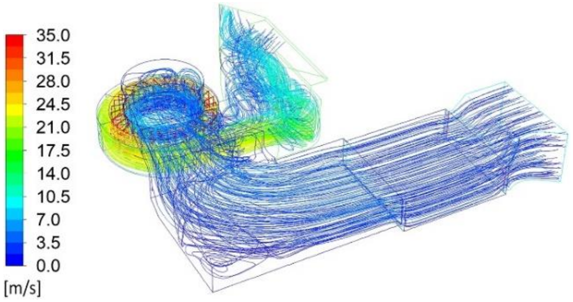

Fig. 13 - Airflow pathlines in exhaust channel after optimization

6.3 Variant study of the outlet diffusers

A separate numerical study was carried out to assess the performance of various types of HVAC diffusers at the outlet to the room. Four different configurations of the ventilation unit were simulated; see Fig. 10: outlet without any diffuser, outlet with a perforated diffuser, outlet with directional vanes, and outlet with three adjustable nozzles. The main aim of the study was to choose the optimal outlet diffuser to:

- improve airflow distribution on the outlet opening (optimally reach uniform airflow velocity at the whole cross-section of the outlet);
- improve the directional characteristics of the airflow to the room (optimally reaching airflow direction normal to the outlet opening).

The change in pressure conditions at the diffuser was reflected in the study by increasing the fan rotation speed in the individual simulations, so the constant volume flow rate of 45 m^3/s was maintained for all four simulated cases.

Fig. 14 shows the pathlines of the airflow at the outlet to the room, for the three cases with HVAC diffusers. On the basis of the variant study, the configuration of the unit with the three nozzles was recommended for further use. This configuration leads to the best airflow distribution in the outlet opening and the best directional characteristics of airflow to the room. Furthermore, the nozzles are adjustable and allow one to change the direction of the air flow according to the user's preference. See also Fig. 15, showing in detail the flow velocity vectors at the outlet from the device to the room.

6.3 Experimental measurement

A physical prototype of the ventilation unit was manufactured, with the optimized geometry, following the numerical study. A measurement of the noise and volume flow rate was conducted under laboratory conditions. The main objective was to confirm that the noise generated by the unit does not exceed the limits given by current regulations and to test the unit under different operating conditions.

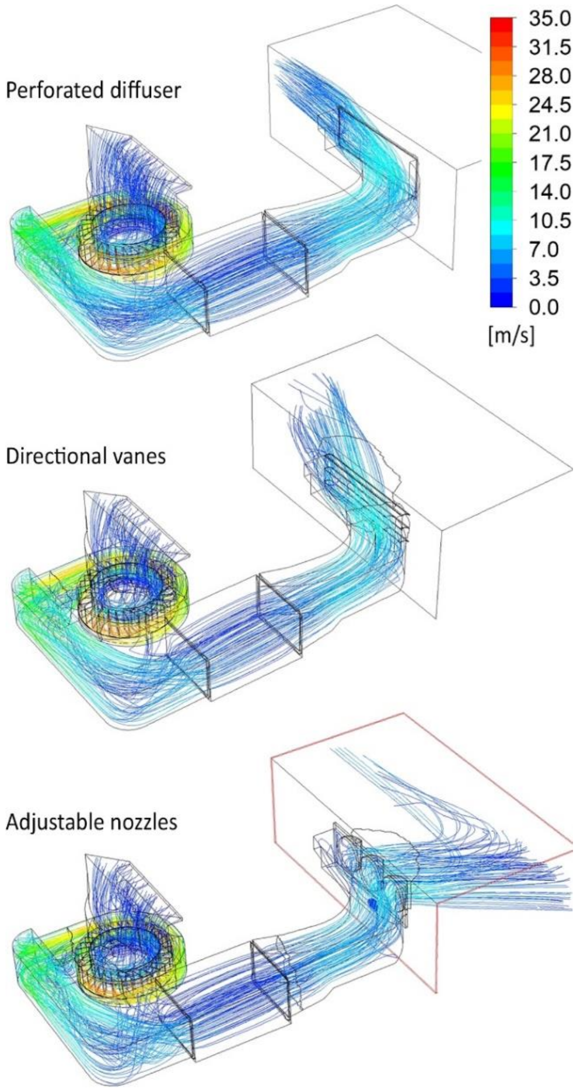

Fig. 14 - Pathlines of airflow to the room – unit equipped with various HVAC diffusers

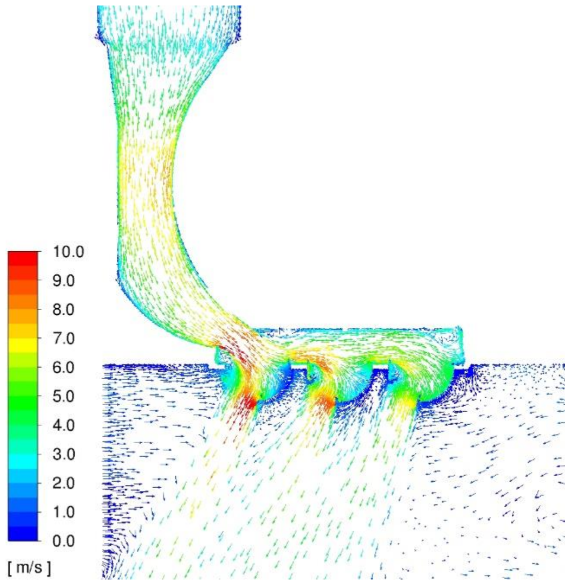

Fig. 15 - Velocity vectors of airflow to the room – unit equipped with adjustable jet nozzles

Following initial measurements in the noise laboratory, the prototype of the unit was further adapted. Selected parts of the sound absorbing material were removed from the walls of the channels. This adjustment had only a small negative effect on the noise attenuation, but the cross section of the channels increased, reducing their pressure losses. Thus, the required airflow was achieved with a lower rotation speed of the fans, which improved the acoustic performance of the device.

It was tested that the ventilation unit reaches the required operational volume flow rate from 15 to 40 m^3/h. It was achieved with the rotation speed of the fans from 1,600 to 3,250 RPM. The corresponding A-weighted levels of acoustic pressure ranged from 15 to 32 dB (measured 3 m from the unit). The values meet the current regulations. It was advised to run the ventilation unit in the low power setting during the night, as it does not exceed the A-weighted level of acoustic pressure 25 dB (measured 1 m from the unit), as requested by the Government Regulation No. 272/2011 on protection of health from adverse effects of noise and vibrations.

6. Conclusion

The main aim of the paper was to demonstrate the practical use of variant numerical analysis for the development of new HVAC appliances. The CFD simulations were used to support the development of a new decentralized compact ventilation unit for local ventilation of rooms.

The numerical model of the ventilation unit was prepared on the basis of the prototype geometry. The Multiple Reference Frame (MRF) method was used to approximate the rotation of the fan impellers. The filtration material was represented in the models by porous media, while the pressure-volume characteristic curve of the filters was prescribed to the model with the use of the PorZo ACT extension.

The initial design of the unit was optimized following the findings of the numerical analysis. Critical points of the prototype geometry were identified and adjusted. These were imperfections in the geometry of the air channels and not optimal orientation of the supply channel fan. It was verified by CFD study that the proposed adjustments were appropriate and that the expected improvements were achieved. In a separate variant CFD study, the use of different types of diffusers at the outlet to the room was addressed. It was recommended to use adjustable nozzles, which allow one to direct the airflow into the room according to the user's preference.

Consequently, it was experimentally verified that the ventilation unit meets the technical requirements. It was tested that the it reaches the operational volume flow rate of 15 to 40 m^3/h with the corresponding A-weighted levels of acoustic pressure in the range from 15 to 32 dB. Thus, the unit meets the noise limits set by current regulations. However, it was advised to run the ventilation unit in low power setting during the night operation, so it does not exceed the requested A-weighted level of acoustic pressure of 25 dB

The outcome of the research is a new type of compact small ventilation unit for local ventilation of rooms with heat and humidity recovery. An increase in device efficiency and a reduction in its acoustic power were achieved, with the help of CFD simulations, while maintaining its compact dimensions.

7. Acknowledgement

This study was supported by the European Union, the European Regional Development Fund, the Operational Program Enterprise and Innovation for Competitiveness.
No. CZ.01.1.02/0.0/0.0/17_107/0012492

8. References

[1] Ryszawy, J. Photos of the ventilation unit, VIC ČVUT, Prague, Czech Republic. 2019

[2] Shih, T., Liou, W., Shabbir, A., Yang, Z., Zhu J. A new eddy viscosity model for high Reynolds number turbulent flows. Computers & Fluids 1995; (24): 227–238.

[3] Zelenský, P., Barták, M., Zavřel, V., Zmrhal, V., Krupa, R. Numerical Analysis of Airflow in a Modular Fan Unit Using CFD Simulation. Proceeding of the 13th REHVA World Congress CLIMA 2019. Bucharest, Romania. 2019.

[4] ANSYS Inc. ANSYS Fluent User‘s Guide. ANSYS Inc. Canonsburg (USA). 2013.

[5] SVS FEM PorZo ACT Extension. Available from: https://www.svsfem.cz/produkty/aplikace-ansys-act/porzo (11. 7. 2020)

[6] Gullberg, P., Sengupta, R. Axial Fan Performance Predictions in CFD, Comparison of MRF and Sliding Mesh with Experiments. Proceeding of the 11th European Conference on Turbo-machinery Fluid Dynamics and Thermo-dynamics. Istanbul, Turkey. 2011.

[7] Gullberg, P., Löfdahl, L., Adelman, S., Nilsson, P. A correction method for stationary fan CFD MRF models. Proceeding of the SAE World Congress & Exhibition. Detroit, USA. 2009.

[8] Moreau, S., Henner, M., Brouckaert, J.F., Neal, D. Numerical and Experimental Investigation of Rotor-Stator Interaction in Automotive Engine Cooling Fan Systems. Proceeding of the 7th European Conference on Turbomachinery Fluid Dynamics and Thermodynamics. Athens, Greece. 2007.