EPJ Web of Conferences **269**, 01056 (2022) https://doi.org/10.1051/epjconf/202226901056
EFM 2019

New ventilation schemes of car cabin evaluated by CFD simulations

Jan Sip[1,*], *Jan* Pokorny[1], *Jan* Fiser[1], *Frantisek* Lizal[1], and *Miroslav* Jicha[1]

[1]Brno University of Technology, Faculty of Mechanical Engineering, Energy institute, Technicka 2896/2, Brno 616 69, Czech Republic

Abstract. Computational fluid dynamics (CFD) was used to investigate new ways of air conditioning of a car cabin during summer conditions. Three different schemes of ventilation were considered: classical mixing ventilation (dashboard vents), large-area ceiling vents, and underfloor vents delivering air under the seats. The Star-CCM+ was used to carry out unsteady RANS simulations and resulting temperature and velocity fields were compared. Furthermore, the draft risk and age of air were evaluated for each scheme. For air conditioning of the cabin, mixing ventilation had the best results.

1 Introduction

Nowadays, the most of car's cabin ventilation system using mixing ventilation (dashboard and floor vents). Different situation is in the case of vans, microbuses, which uses ceiling vents. The benefit of this more flexible interior (removable seats etc.). With the developments of autonomous car and flexible interiors the convectional solution of cabin ventilation became restrictive and there are posibilies for new concepts and desings. In the case of the flexible interior, it is assumed that the front seats can be rotated in the opposite direction of travel, so the convectional ventilation system loses its efficiency. From this perspective, the research into new ventilation systems is important.

The new concepts of autonomous vehicle ventilation system has already been investigated experimentally and computationally by Dehne [1]. Dehne has been tested three new modes of vertical ventilation: air intake under the seats, large-area diffusers in the roof of the car, and the combination of both. Inside the cabin there were sitting four manikins, and the cabin was fitted with a number of air velocity, temperature and relative humidity probes. For the CFD simulation the OpenFoam was used, just computational mesh was created in the commercial software Star-CCM+ with base size of cell 4 mm. The turbulence model k-ω SST and. the "buoyantBoussinsq" model were used. From this study vertical ventilation has many advantages with combined air supply achieved the best results. However, the effect of direct solar radiation was not included in the study.

In the presented study we investigated summer test case (including the solar radiation) when three different variant of cabin cooling were simulated. The criteria for each variant was the value of the draft risk and age of air were evaluated with the focus on the driver seat. These simulations help to identify behaviour of the ventilation system in the early phase of mock-up design process before its own construction, which will follow.

2 Materials and methods

As first the concept of cabin mock-up was sketched and than 3D stl geometry created using CAD. For the CFD simulation the Star-CCM+ ver. 10.04 was used. Own simulations were performed on the virtual mock-up using computational cluster at NETME centre.

2.1 Geometry and mesh

Three dimensional geometry of the mock-up (**fig. 1**) was created in terms of Josef Bozek Competence for

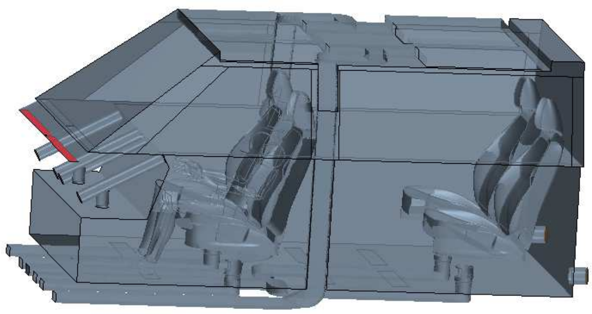

Fig. 1. Geometry.

Automotive Industry The mock-up represents just the cabin without hood and width and height of the cabin corresponds to the typical van with the dimensions 3×1.4×1.38 m (just the length was reduced from 4.5 m to 3 m, we not considered the 3rd row of seats). The total air volume of cabin is 5.7 m^3. In the cabin there are four seats

* Corresponding author: jan.sip@vutbr.cz

and one manikin at the driver‘s seat. The polyhedral computational mesh was used with base size 0.025 m and with the total number of cells 4.5 million. All parts of the geometry except the supply ducting contained five prism layers. The surface mesh of the manikin and the seats was more refined to Relative Target Size 0.0075 m. Furthermore, the volume mesh was refined locally in the vicinity of active vents.

Table 1 HVAC boundary conditions

Type	Label	Velocity inlet [m/s]	Temperature [°C]
Mix	D1	4.17	12.5
	D2	4.28	13.0
	D3	4.60	11.5
	D4	4.24	11.5
	Def1	0.17	18.8
	Def2	0.33	18.8
Floor	F1 F2 F3 F4	4.14	12.1
Ceiling	C1 C2	8.29	12.1

2.2 Physical models

The simulations were performed as unsteady with timestep 0.001 s, and Courant number of 5. The temporal discretization was performed by the implicit unsteady algorithm with the second-order temporal discretization scheme, and the convection term was discretized using the second-order upwind scheme. For the radiation modelling the view factors were calculated and the surface to surface model was used. The buoyancy, Ideal gass and solar loads in the cabin were taken into account. Turbulence was modelled by the k-ω SST model.

2.3 Boundary condition

Environmental

For summer test case was important to consider Multiband Thermal Radiation model which splits radiation spectrum into shortwave (0,01 - 2,5 μm) and longwave (2.5 - 1000 μm). The value of the solar loads was 800 W/m2 with a 90° elevation angle. The ambient temperature was 30 °C.

Cabin

The material properties of mock up were defined based on the properties of real silver painted car with tinted window, see [2, 3]. The cabin boundary conditions of the wall type were set on the car parts. The surface temperature, the thermal resistance of the material and the heat transfer coefficient were defined. Emissivity and transmissivity of the materials were defined. Transmissivity for both type of radiation was prescribed.

HVAC

The volumetric flow rate for individual ventilations modes was constant (99 l/s). Two boundary conditions of the velocity inlet type are prescribed for ceiling ventilation (CV). The inlet air speed is 8.3 m/s and the air temperature is 12.1 °C. Four identical boundary conditions were prescribed for floor ventilation (FV). The inlet air speed is 4.1 m/s and the air temperature is 12.1 °C. An overview of boundary conditions for mixing ventilation (MV) is in the **table 1**.

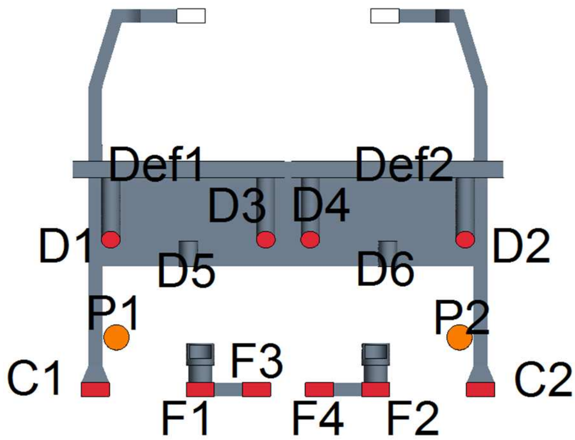

Fig. 2 Inlets.

Manikin

The surface temperature was prescribed 34 °C on non-clother parts of the manikin. The skin temperature was defined as 34 °C and the temperature resistance of the clothing related to the summer period on the parts of the manikin on which the clothing occurs.

2.4. Post-processing and outputs

The comparison was made between flow velocities and temperature of air at the monitoring points at important points (ankle, hand, head). Furthermore, the Draft risk was elavuated by ISO 7730 [3] as:

$$DR = (34 - t_A)(u - 0.05)^{0.62}(0.37u\sqrt{\frac{2}{3}k} + 3.14) \quad (1)$$

where t_A is the air temperature (°C), u is the velocity magnitude (m/s) and k is the turbulent kinetic energy (m^2/s^2). The part $\sqrt{\frac{2}{3}k}$ is the turbulence intensity. Criteria for index DR can be split into Category A (DR 10 %), Category B (DR 20 %), Category C (DR 30 %).

Age of air (s) is definited:

$$\tau_i = \int_{t_1}^{t_2} \left[1 - \frac{c_i(t_1)}{c_i(t_2)}\right] dt \quad (2)$$

where C_i is the contaminant concetration (kg_c/kg_s) and t is the time (s).

3 Results

The results of the numerical simulations are presented as mean scalar field Age of air (AA) and Draft risk (DR). The object of interest was mainly the plane going through the centre of the driver.

The average AA fields in **fig. 3** also show that the freshest air near the driver's face is in the case of FV, MV, and the oldest air is in the case of CV.

In terms of DR, the best results were provided by the MV type, where there are no higher DR values near the driver. For type CV, the DR values in the thigh area are high, but the sensitivity to DR is lower thanks to the clothing layer.

In the FV type, there are very high draft risk values in the calf area, which are expected to have a summer case, their high susceptibility to DR.

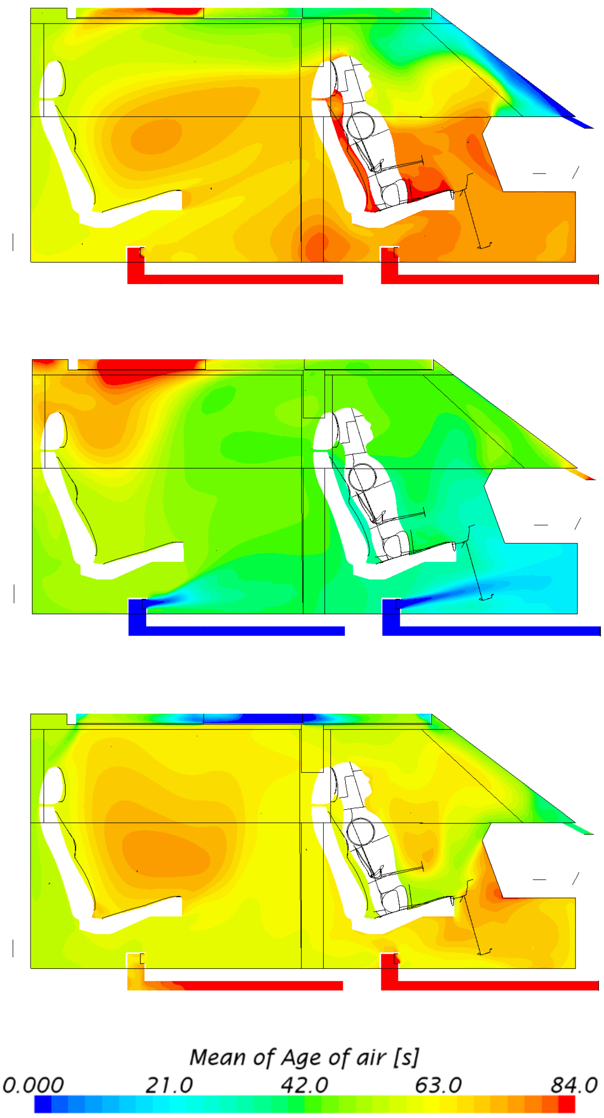

Fig. 3. Mean of Age of air (MV,FV,CV).

Table 2. Time averaged temperature, velocity magnitude, AA and DR in point at driver's nose.

	T [°C]	U [m/s]	AA [s]	DR [%]
Mix	25.45	0.178	52.9	7.8
Floor	22.17	0.148	45.2	10.4
Ceiling	26.85	0.163	65.7	6.1

The **table 2** shows the time averaged values of temperature, velocity magnitude, AA and DR in point near driver's nose. The highest temperature at the monitoring point is in the case CV, while the lowest in the case FV. The speeds for individual types of ventilation are up to 0.2 m/s. The highest AA is in case of CV and the lowest in the case FV. The maximum DR value is for type FV. Other types have DR up to 10 %.

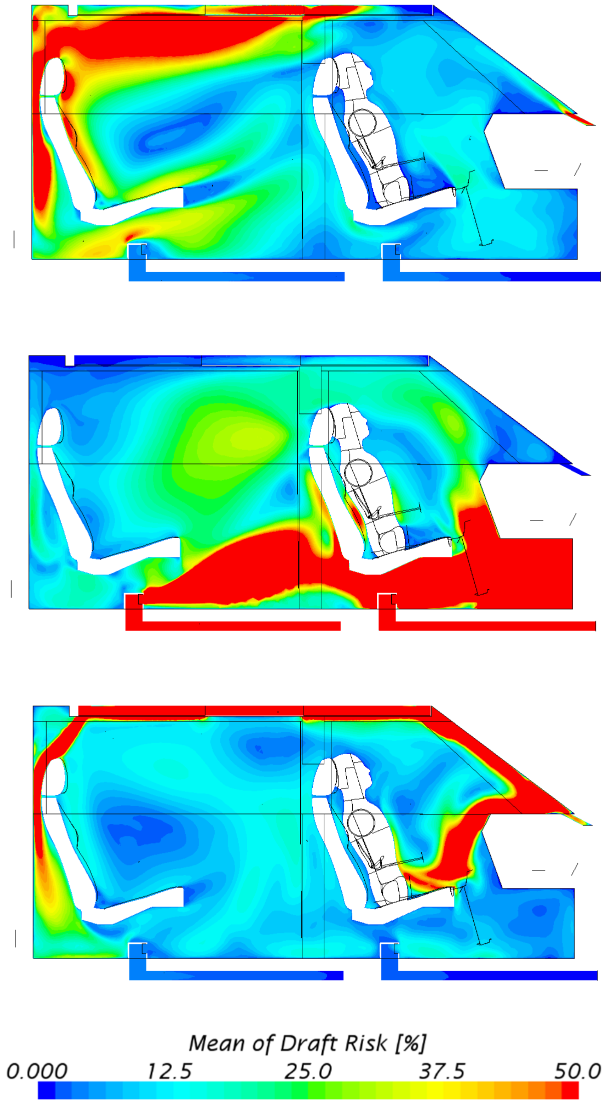

Fig. 4. Mean of Draft risk (MV,FV,CV).

From the results it is also evident that in the type of ventilation MV open ceilings vents in the flowing air, which leaves them in the rear of the car and can have a negative effect on the passenger. In the CV type, the main current core of the large-area vents diffusers adheres to the front and rear window respectively. This is due to the Coanda effect.

4 Discussion

In this first study just the classical interior was investigated.
MV is the best type for driver's comfort in terms of DR index. This is followed by CV and FV.
MV also provides the best results of AA, followed by CV and FV.

The mixing ventilation was evaluated as the optimal, however in case of flexi interior we can expect that the new designs of ventilation system will be more beneficial. To evaluate whole cabin is not presented in this study because should be considered effect of manikins sitting on rear seat

If we wanted to determine AA and DR for other passengers, they would have to be included in the simulation, because the presence of the manikin has a substantial influence on the flow field.
However, from the results (only with a manikin on the driver's seat), a higher DR level in the rear seat behind the driver can be included. In the FV type, it is possible to deduce a higher DR level in the rear seat passenger at the calf area.

5 Conclusion

The virtual mock-up was designed and the CFD simulation of internal air flow were performed with focus on temperature and velocity field Furthermore the index age of air and draft risk were evaluated especially in vicinity of manikin.
From obtained results can be concluded:

- The geometry of the manikin has a great influence on the flow pattern inside vehicles, so there is no point in evaluating the thermal comfort values in the seat where the manikin is not located.
- In mixing ventilation case large-area ceiling vents should be closed to avoid massive flow through ceiling channels allows to In the next step, these vents will be defined as a porous baffle, which should eliminate this effect.

For air conditioning of the cabin in summer conditions the MV had the optimal AA and DR.
The next step is to optimize ratio between flow rates from FV and CV and carried out evaluation of thermal comfort using equivalent homogenous temperature.

Acknowledgement

This research has been realized using the support of Technological Agency, Czech Republic, programme National Competence Centres, project # TN01000026 Josef Bozek National Center of Competence for Surface Vehicles. This support is gratefully acknowledged.

References

1. T. Dehne, P. Lange, A. Volkmanns, Building and environment, **129** (2018)
2. C. Kučera, Mater Diploma Thesis, Brno University of Technology, 2018
3. V. Pavlík, Master Diploma Thesis, Brno University of Technology, 2018