

Numerical Investigations on Indoor Environment in Vehicles

A. Ene and C. Teodosiu

Abstract— The transition from automobiles using petrol as main fuel for transportation to full electric vehicles is rocketing nowadays. Hence, the impact of energy consumption of the auxiliary systems in a vehicle plays a key role in this subject as it drastically influences the driving range, as well as lowering the engine's power. The HVAC equipment is the most energy consuming auxiliary system in vehicles. Therefore, an efficient operation of the HVAC system is essential when talking about electric vehicles. This paper evaluates the parameters such as velocity, temperature and humidity levels inside a vehicle's cabin based on CFD numerical studies. The 3D geometrical characteristics were developed based on a real 1:1 scale experimental stand presenting the interior dimensions of a vehicle which was placed inside a climatic chamber. This study provides an analysis of the parameters during winter conditions as part of a broader subject of numerical analysis of the condensation phenomena taking place inside vehicles.

Keywords—CFD modeling, indoor environment vehicles

I. INTRODUCTION

The evolution and transition toward vehicles using electricity as main fuel for transportation gained momentum within the last years, on one hand due to the development of the types of energy production as well as due to the evolution in material production which allowed to this technology the possibility to be implemented at a wider scale. On the other hand, this transition started to be more and more important as a response to the environmental changes the entire world is facing today [1].

Nevertheless, the autonomy of a full electrical vehicle is strongly influenced by the energy consumption of auxiliary systems, such as the HVAC system. In comparison with the normal cars using petrol as main fuel for transportation, the electric vehicles do not produce sufficient waste energy that can be used to raise the comfort conditions while driving without any impact on the autonomy of the driving range. Therefore, for cold climate regions, the HVAC usage can result in a drop in the autonomy of the vehicle up to 50% [2]. The classic vehicles using traditional fuel dispose of approximately 5kW residual heat, whilst the full electric vehicles produce only 2 kW

residual heat, which is insufficient to maintain the indoor air parameters at comfortable conditions [3].

Taking into consideration the arguments above, there is a strong need to lower the energy consumption of the auxiliary systems in vehicles. This can be done through several approaches. This paper proposes to present the indoor parameters inside a vehicle's cabin and their future influence on the energy consumption.

The evaluation of the energy consumption inside a vehicle's cabin is not as simple as the one in buildings. The interior volume of a cabin which is small in comparison with the volume of a normal room, as well as the non-uniformity of the temperature of the interior space, the glazed surfaces, direct solar radiation and its' probability to rapidly change lead to a more difficult approach to evaluate the enclosed environment of a vehicle [4]. This paper proposes a numerical approach to determine the thermal comfort factors inside a vehicle based on a real 1:1 scale experimental stand. This work is part of a broader study dealing with the condensation phenomenon inside vehicles, especially on the windshield area which can highly impact the driving security of the passengers and energy consumption for demisting

II. EXPERIMENTAL SET-UP

A. Climatic chamber

The first step in this work was represented by the construction of the vehicle's cabin. The construction was developed inside a climatic chamber which can be found at the Faculty of Building Services at the Technical University of Civil Engineering in Bucharest. For this stage, the interior cabin of a real vehicle was taken into account at a 1:1 scale. It is worthwhile to mention that special attention was paid to details and interior equipment of the cabin car. Consequently, the real configuration of a car was implemented as much as possible, including the windshield, the dashboard together with all the interior original vents and distribution pipes, the steering wheel, as well as the sitting chairs and benches (Fig. 1). Furthermore, this construction was placed inside the climatic chamber in order to evaluate the influence of the outside parameters on the vehicle's indoor environment.

Manuscript received May 17, 2022.

A. E. Author is a Ph.D. student at Technical University of Civil Engineering Bucharest.

C.T. Author is a professor taking part at the department of thermohydraulic systems and equipment for atmosphere protection from Technical University of Civil Engineering Bucharest.



(a)



(b)

Fig. 1. Experimental set-up

III. NUMERICAL APPROACH

A. Description of the numerical modelling

The experimental stand represented the base for the further numerical studies. The geometrical characteristics of the real model were implemented in a 3D representation using SolidWorks. Further on, the geometrical model was imported in Ansys FLUENT where it was prepared for the next stages of the numerical studies (Fig. 2).

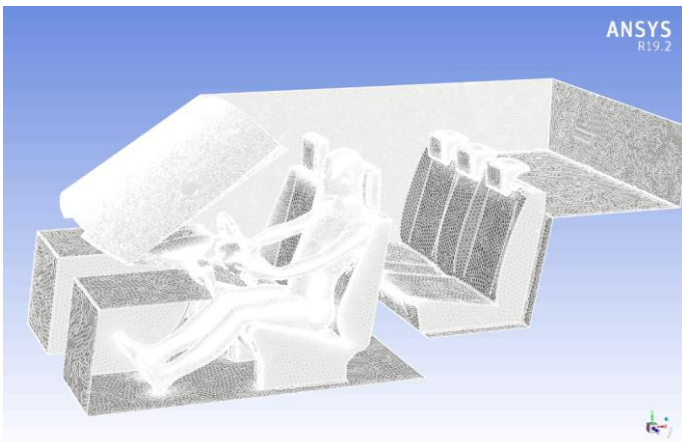


Fig. 2. 3D geometry of the vehicle's cabin

Several types of discretization models were analyzed, in the

end resulting a mesh of approximately 8.58 million cells. This mesh was tested and represented the best option for accuracy and calculation time.

The main characteristics of the proposed CFD model are shown in Table I. This numerical model was built using the general-purpose, finite-volume, Navier-Stokes solver - Fluent version 19.2.

TABLE I. CFD MODEL - PRINCIPAL ELEMENTS AND HYPOTHESIS

Fluid	Air – water vapor mixture
Flow	Three-dimensional, steady, non-isothermal, turbulent
Computational domain discretization	Finite volumes, unstructured mesh (tetrahedral elements): 8 580 763
Turbulence model	Shear Stress Transport (SST) turbulent kinetic energy-specific turbulent dissipation rate (k- ω), with low-Reynolds corrections
Numerical resolution	Second-order upwind scheme Velocity-pressure coupling: coupled algorithm Convergence acceleration: algebraic multigrid

B. Studied cases

For the simulations, it was considered the winter conditions in Bucharest, Romania (humid continental climate). Practically, it was considered an exterior temperature of 0 °C and water vapors content of 0.8 g/kg (according to mean weather winter conditions in Bucharest). For the ventilation system inside the vehicle, the demisting grids were used, which introduced an exterior air with a speed of 12.5 m/s at a temperature of 30 °C.

In order to integrate in CFD simulations the humidity, an equation for the conservation of the water vapor is added to the main equations of the CFD model. This equation is similar to classical transport CFD equations, taking into account transport and diffusion (molecular and turbulent) phenomena for the water vapor [5]. In addition, it was considered a humidity release from the driver of 3.26×10^{-5} kg/s [6] and a heat release of 80 W [6]. The numerical studies were developed in order to determine the impact of the imposed boundary conditions on the final results. As a result, three different situations were analyzed:

- the first case proposed to impose convection heat transfer coefficient only for the windshield area;
- the second case proposed to impose superficial heat transfer coefficients for all exterior surfaces of the car;
- the third case imposed a radiation model together with the heat transfer coefficients for all the exterior surfaces.

The heat transfer coefficients were found in the literature and were chosen as can be seen in table 2.

TABLE II: HEAT TRANSFER COEFFICIENTS FOR CONVECTION [7]

	Stationary conditions
Windshield	20 W/m ² K
Lateral windows	15 W/m ² K
Ceiling	3 W/m ² K

With regards to the radiation model, a parallel between the Surface-to-Surface (S2S) model and Discrete Ordinates (DO) model, both available in Fluent was started. The S2S model does not take into consideration the interior environment as participating in the radiation heat transfer, whilst the Discrete Ordinate (DO) model does. On the other hand, the S2S model is less time consuming and does not require as many computational resources as the DO model [8]. Although it is known that water vapors can have an influence over the radiation heat transfer [9], the majority of the numerical studies neglect their implication as it was proven to be insignificant for heat transfer in enclosed environments [10]. Moreover, as the water vapors concentrations are low for the interior volume of a vehicle and the concentration of carbon dioxide is small, the air is generally considered as a transparent environment in indoor spaces [11]. Therefore, for this analysis the S2S model was used.

The results were analyzed based on six different points inserted inside the vehicle as can be seen in Figure 3 (a and b). Point P1 corresponds to the region near the ventilation grid, whilst points P2 and P3 were placed in the occupants’ region in the front of the car, points P4 and P5 are corresponding to the passengers who would have been placed at the back of the car, whilst point P6 was placed in the middle of the cabin, between the two front chairs.

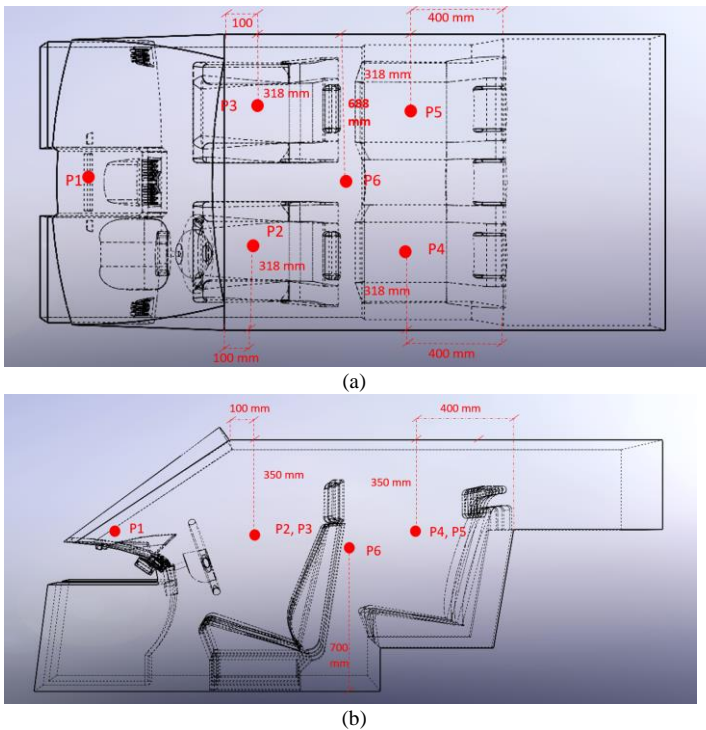


Fig. 3. Measurement points

IV. RESULTS

The first case showed that after approximately 4000 iterations, the interior temperature reached a stable temperature of approximately 26 – 28 °C depending on the position of the

point as can be seen in Fig. 4.

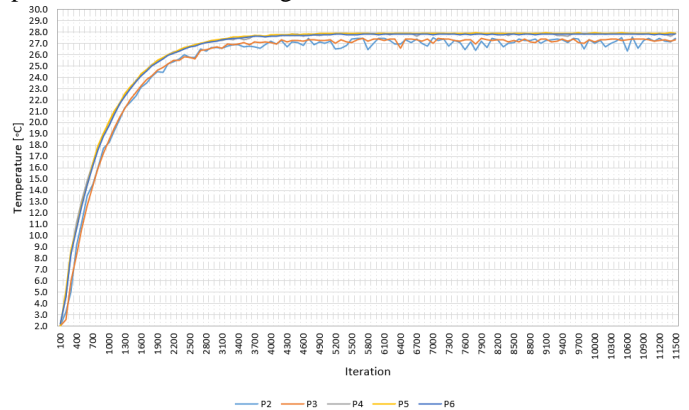


Fig. 4. Temperature for points P2 – P6, case 1

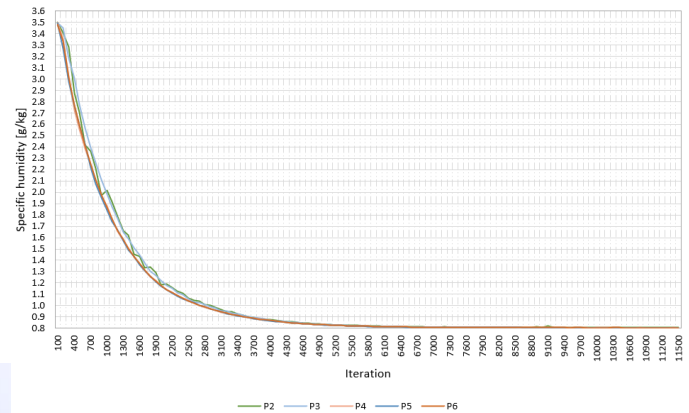


Fig. 5. Humidity levels for points P2 – P6, case 1

In addition, Fig. 5 shows that after approximately 6100 iterations the interior humidity levels are starting to stabilize at a value similar to the one introduced from the exterior air.

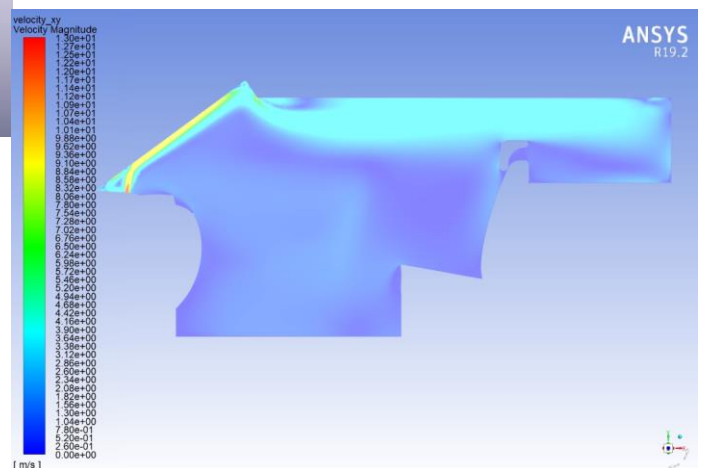


Fig. 6. Velocity contour, case 1

Fig. 6 presents the velocity contour for the first case

Next, the contours for different thermal comfort factors inside the vehicles were analyzed. These contours were all related to the median longitudinal section through the vehicle.

Fig. 6 emphasizes the velocity values inside the cabin when the demisting grids are being used. The air jet presents a higher

value near the air introduction grid region of 12.5 m/s. The air jet further takes the form of the windshield dropping at a value of about 9.2 m/s and then develops towards the back of the car at the ceiling level at a velocity of around 3.3 m/s.

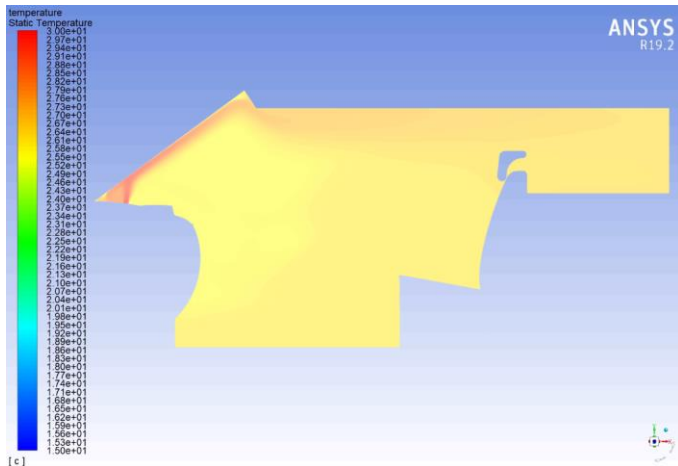


Fig. 7. Temperature contour, case 1

Fig. 7 presents the interior temperature for the first case. It can be seen that the interior temperature presents a uniform distribution with a value around 26 °C with a higher value for the region where the air is introduced inside the cabin which is around 30 °C.

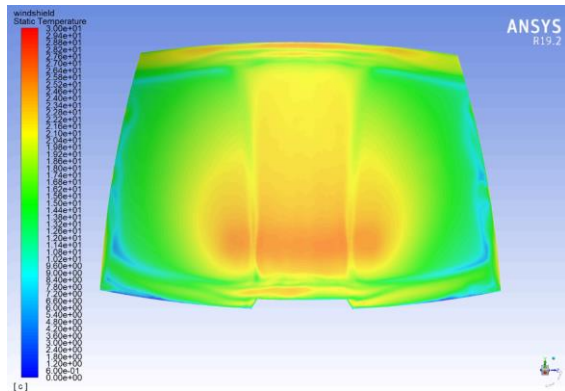


Fig. 8. Windshield temperature, case 1

The temperature at the windshield region can be seen in Fig. 8. This image shows lower temperature values for the outer regions where they drop to values of around 3°C, whilst the central region which receives the direct air jet from the internal grids, presents a value of around 24°C.

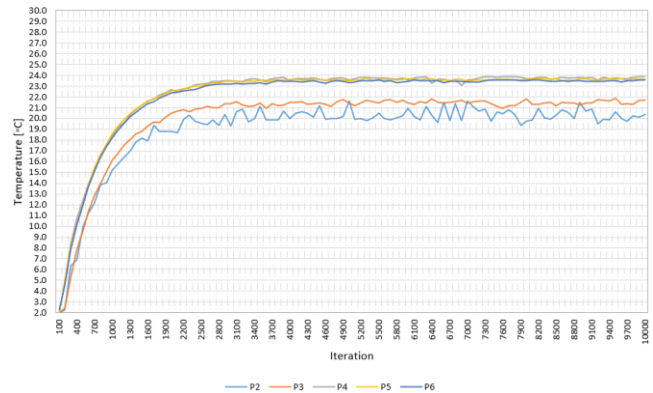


Fig. 9. Temperature for points P2 – P6, case 2

The second case showed a drop in the air temperature for the points P2 – P6. If the first case showed values of around 27 °C, this case highlights values of around 24 °C. These values are in accordance with the heat transfer coefficients which were applied for all exterior surfaces of the cabin.

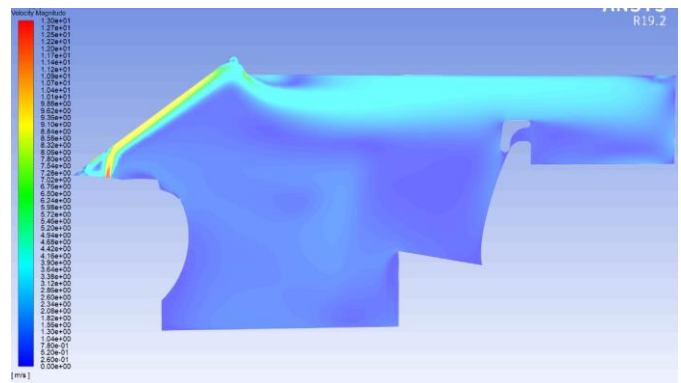


Fig. 10. Velocity contour, case 2

Fig. 10 shows the velocity contour for the second case which does not prove any particular differences in comparison with the first case. This proves that the inertial forces have a more important influence in comparison with the buoyant forces.

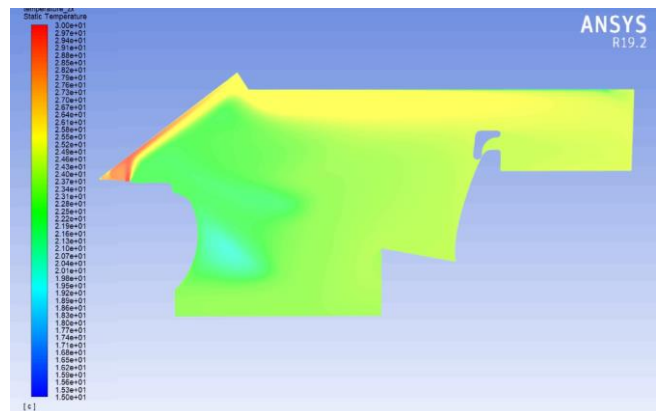


Fig. 11. Temperature contour, case 2

Fig. 11 shows the temperature contour for the second case. This case shows significant differences in comparison with the first case. Due to the implementation of heat transfer coefficient for all the exterior surfaces, the temperature drops inside the vehicle's cabin, especially in the front part. As the warm air jet

goes towards the back of the car, as can be seen in Fig. 11, the driver and the front passenger experience lower temperatures of around 22°C.

The results of the third case (taking into account the S2S radiation model) are presented in Fig. 12. It can be noticed that after around 2800 iterations, the interior temperature started to present constant values for the P2 – P6 points. In comparison with the second case, it can be seen that interior temperature drops to a lower value of around 23 °C.

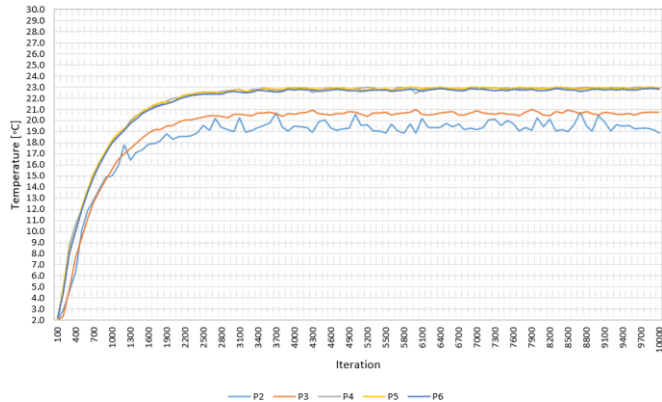


Fig. 12. Temperature for points P2 – P6, case 3

As the velocity contour remains the same for the third case, it was not presented once again. Therefore, only the temperature contour can be seen in the next figure.

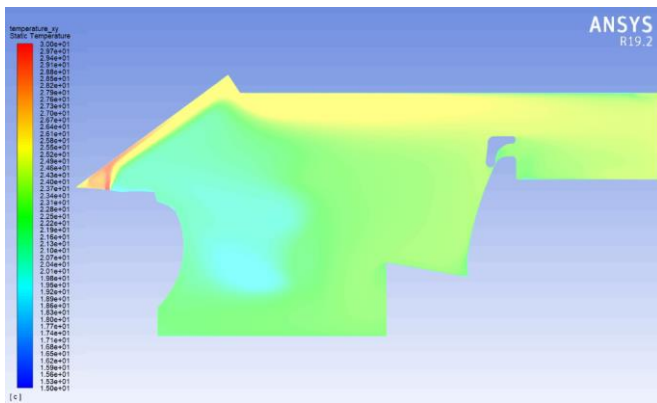


Fig. 13. Temperature contour, case 3

In comparison with the second case where the thermal radiation was not taken under consideration, it can be seen that the temperatures inside the vehicle are more uniform (Fig. 13) now. Furthermore, the temperature for the front part of the vehicle presents a value of around 19 °C in this third case in comparison with the 23 °C value met in second case.

Fig. 14 presents the temperatures at the windshield level. This figure shows an important temperature drop in comparison with the previous case. Although the convective heat transfer remains the same for all three cases, the influence of the heat transfer by radiation with other car surfaces is important and leads to a drop in the temperatures at the windshield level.

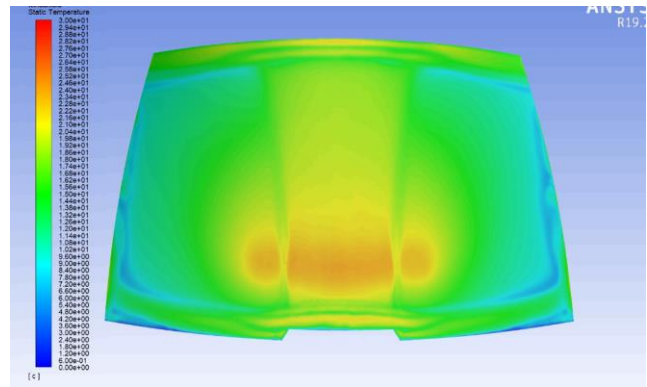


Fig. 14. Temperature distribution for windshield, case 3

V. CONCLUSION

This study aimed to determine the parameters inside a vehicle’s cabin during cold periods when the demisting system is used. This study is part of a boarder Ph.D. analysis for determining the condensation phenomena inside a vehicle, especially at the windshield level.

The present study showed that the air velocity inside a vehicle’s cabin does not suffer huge modifications when the exterior convection heat transfer coefficients are modified. This phenomenon proves that the inertial forces are influencing the flow and that they are more important in comparison with the forces generated by differences in temperature and humidity. Furthermore, it was shown that the temperature inside the cabin presents lower values for the front passengers’ part in comparison with the back part of the car when using the demisting/ de-icing air grids. This phenomenon took place for all the three analyzed cases, even though the temperatures were slightly different. Also, it is worth mentioning that this phenomenon is in correlation with the air velocity distribution.

Another important aspect of this study is represented by the impact of the thermal radiation on the final results. It was demonstrated that the results are highly influenced by the radiation heat transfer. This influence can be correlated for point P6 corresponding to the middle of the vehicle. For this point it can be seen that the temperature drops to 22.8 °C (case 3) from 23.5 °C (case 2). This influence was entirely due to the radiation heat transfer.

Finally, this study highlights the importance of imposing the appropriate boundary conditions starting from an incipient phase as they highly influence the final results of the numerical modelling. This paper shows that by imposing a heat transfer for all the exterior surfaces of the vehicle leads to serious impact on the temperature at the windshield level. Moreover, the heat radiation transfer showed an improvement on the uniformity of the air temperatures. This phenomenon has a massive influence on the further studies which will consist in analyzing the condensation phenomenon inside vehicles, especially on the windshield. This situation will be further evaluated through a numerical simulation.

The study presents the perspective of further evaluation of the interior conditions determined through an experimental campaign, thus providing the possibility to compare and

validate the numerical results.

REFERENCES

[1] Z. Zhang, D. Wang, C. Zhang, and J. Chen, "Electric vehicle range extension strategies based on improved AC system in cold climate – A review," *International Journal of Refrigeration*, vol. 88, pp. 141-150, 2018.
<https://doi.org/10.1016/j.ijrefrig.2017.12.018>

[2] D. Leighton, "Combined Fluid Loop Thermal Management for Electric Drive Vehicle Range Improvement," *SAE International Journal of Passenger Cars - Mechanical Systems*, vol. 8, no. 2, pp. 711-720, 2015.
<https://doi.org/10.4271/2015-01-1709>

[3] Z. Qi, "Advances on air conditioning and heat pump system in electric vehicles – A review," *Renewable and Sustainable Energy Reviews*, vol. 38, pp. 754-764, 2014.
<https://doi.org/10.1016/j.rser.2014.07.038>

[4] S. Yun *et al.*, "Prediction of thermal comfort of female passengers in a vehicle based on an outdoor experiment," *Energy and Buildings*, vol. 248, 2021.
<https://doi.org/10.1016/j.enbuild.2021.111161>

[5] R. Teodosiu, "Integrated moisture (including condensation) – Energy–airflow model within enclosures. Experimental validation," *Building and Environment*, vol. 61, pp. 197-209, 2013.
<https://doi.org/10.1016/j.buildenv.2012.12.011>

[6] J. Liu, H. Zou, G. Zhang, X. Zhang, and C. Tian, "Analysis of Maximum Return Air Ratio on No-fogging Condition by In-cabin Climate Simulation for EV," *Energy Procedia*, vol. 105, pp. 2445-2450, 2017.
<https://doi.org/10.1016/j.egypro.2017.03.703>

[7] Y. Mao, J. Wang, and J. Li, "Experimental and numerical study of air flow and temperature variations in an electric vehicle cabin during cooling and heating," *Applied Thermal Engineering*, vol. 137, pp. 356-367, 2018.
<https://doi.org/10.1016/j.applthermaleng.2018.03.099>

[8] "ANSYS Fluent Theory Guide " 2015.

[9] X. Li and J. Tu, "Evaluation of the eddy viscosity turbulence models for the simulation of convection–radiation coupled heat transfer in indoor environment," *Energy and Buildings*, vol. 184, pp. 8-18, 2019.
<https://doi.org/10.1016/j.enbuild.2018.11.043>

[10] T. Kogawa, L. Chen, J. Okajima, A. Sakurai, A. Komiya, and S. Maruyama, "Effects of concentration of participating media on turbulent natural convection in cubic cavity," *Applied Thermal Engineering*, vol. 131, pp. 141-149, 2018.
<https://doi.org/10.1016/j.applthermaleng.2017.11.135>

[11] X. Li, Y. Yan, and J. Tu, "Evaluation of models and methods to simulate thermal radiation in indoor spaces," *Building and Environment*, vol. 144, pp. 259-267, 2018.
<https://doi.org/10.1016/j.buildenv.2018.08.033>



Alexandra Ene received a Bachelor’s degree at the Technical University of Civil Engineering in Bucharest at the Faculty of Building Services. She further continued her Masters’ Degree at the same university with the specialization of Energy Efficiency of Building Services in Constructions. Currently, she is enrolled as a Ph.D. student at the Doctoral School at TUCEB.

She participated in several student competitions such as REHVA Student Competition in 2019, Bruxelles, Belgium where she won the third place and AIIR Student Competition where she was rewarded with the first prize. She participated as a Decathlete with the Romanian team at the Solar Decathlon Europe 2019, Szentendre, Hungary where they won together the third place. Her current research areas are related to CFD modelling, energy consumption of buildings and building services as well as comfort conditions and occupancy behaviors.



Cătălin Teodosiu, born in Bucharest, Romania, July 22, 1972, received the B.E. degree in Civil Engineering (1996), the M. Sc. degree in Energy Management (1997) from the Technical University of Civil Engineering (Building Services Faculty), Bucharest, Romania and the Ph.D. degree in Civil Engineering (2001) from the National Institute of Applied Science (INSA) of Lyon, France. Between 1996-1998 (Assistant Professor) and from 2004 till now (Lecturer and Associate Professor), he joined the Thermo-Hydraulic and Protection of the Atmosphere Systems Department, Faculty of Building Services and Equipment, Technical University of Civil Engineering, Bucharest.

He performed teaching and research activity also in France, Assistant Lecturer and Researcher, 1998–2004 (University Claude Bernard – Lyon I, INSA of Lyon and CETHIL – The Center for Thermal Sciences of Lyon). He is author and co-author of 4 books and university courses. He published over 100 articles in international and national journals and papers in peer-reviewed international conferences proceedings. He was participant at 8 international research-development-innovation projects (4 as project manager); 1 national research project financed by European Union; 25 national research-development-innovation projects (2 as project responsible). He is also co-author at 2 national technical regulatory documents. He is energy building auditor (Ministry of Regional Development and Tourism – MDRT authorization) and technical expert for HVAC installations (MDRT authorization). His current research interests include CFD (Computational Fluid Dynamics) modeling, focusing on turbulence models and integrated heat-airflow-moisture models; building simulation; high efficiency buildings.