Article citation info:
JANICKA, A., ZAWIŚLAK, M., GŁOGOZA, A. et al. Optimization of the electric bus radiator design in terms of noise emissions and energy consumption by computational fluid dynamics. *Combustion Engines*. 2022, **191**(4), 41-50. https://doi.org/10.19206/CE-147256

Anna JANICKA
Maciej ZAWIŚLAK
Artur GŁOGOZA
Radosław WŁOSTOWSKI

Combustion Engines
Polish Scientific Society of Combustion Engines

Optimization of the electric bus radiator design in terms of noise emissions and energy consumption by computational fluid dynamics

ARTICLE INFO

Received: 19 January 2022
Revised: 7 March 2022
Accepted: 9 March 2022
Available online: 13 March 2022

The paper presents the numerical optimization of an innovative radiator for use in electric buses in terms of energy consumption and noise emission. Computational fluid dynamics simulations were performed. The flow of the cooling medium was modeled using the RANS method. The two-equation k-ε turbulence model, the heat transfer model and the acoustic model were used. According to the research results, the separation of the air stream in individual fan sections contributes to the improvement of energy efficiency and reduces noise emissions. As a result of the simulation, it was found that the best solution in terms of noise emission as well as the occurring flow phenomena caused about a 2 dB decrease of maximum values of the noise level and allowed the equalization of the cooling medium velocity (prevailing velocity range between 4 and 9 m/s). The results of the simulations were verified under laboratory and field conditions, showing a very good convergence of the model with the results of the experiments (i.e. the maximum noise level was estimated at 57 dB, under measurement conditions for the same operating point at 59 dB) while maintaining the baseline energy demand, which indicates a new approach in the method of shaping internal elements of electric vehicle coolers.

Key words: *automotive engineering, electric bus, CFD, noise, ecodesign*

1. Introduction

The dynamic development of electromobility poses challenges to designers regarding not only the efficiency of energy transformation but also the battery life, which is influenced by the stability of its operating temperature. Sustainable design of cooling systems requires optimization of energy management taking into account other important environmental parameters. The multifaceted ecodesign of motor vehicles concerns not only reduction and optimization of energy and material consumption as well as pollutant emissions, but also aspects related to vibration and noise protection for both vehicle users and the external environment. The structure of the motor vehicle market is changing dynamically. In addition to the challenges common to all types of vehicle, such as aerodynamics and body morphology, ecodesign varies greatly depending on the type of drive used.

In the case of conventional vehicles equipped with internal combustion engines (ICE), designing in accordance with the principles of sustainable development currently focuses mainly on rightsizing trends [5], improving the efficiency of fuel combustion and its composition, as well as the efficiency of exhaust gas treatment systems [23, 24]. An interesting approach to improving ICE efficiency is demonstrated in the work of Sroka and Sadlak [18], where the authors present study results showing that the use of an active combustion chamber (based on thermal barriers) can improve the effective efficiency of the engine from 10 to 20%.

However, the dynamic development of electromobility observed in the last decade poses further challenges for designers. In addition to the issue of energy transformation efficiency, lifetime, capacity and recycling of batteries, there are also areas of concern identical to the vehicles equipped with a classic drive, such as noise emission. In this case, nevertheless, it does not apply to the drive unit itself, i.e. the electric motor, but to the coexisting systems. The use of an electric motor both as a stand-alone drive and in a hybrid system requires the use of new solutions in the field of cooling systems, whose task is to maintain the vehicle's batteries in appropriate thermal conditions.

The thermoregulation systems of batteries for use in electric vehicles are currently the subject of numerous studies related to energy optimization, the type and composition of the coolant, their effectiveness, noise emission and control. Shashank [15], in his literature review, emphasized the importance of thermal management in electric vehicles and its impact on the life of lithium-ion batteries, which are significantly sensitive to temperature. A year later, a similar review paper was published by Kim [11], demonstrating the advantages and disadvantages of various battery thermal management system (BTMS) solutions and presenting his own solution in this area. The dynamic development of technology and the ever-growing interest in battery cooling are also indicated in a paper published the following year. Mengyao et al. [12] demonstrated advances in research on electric vehicle cooling technology aimed at eliminating temperature peaks and large temperature gradients. He presented an overview of existing battery thermal management systems and the advantages and disadvantages of cooling systems based on both air and liquid cooling. He defined the trends in their development, such as miniaturization or reduction of relatively high energy demand. The paper also indicated that minimizing the impact on user comfort (mainly in relation to the noise generated by the airflow through the fan and enclosure) is an important research issue.

The problem of research on BTMS was also highlighted by Akinlabi and Solyali [1], who identified the incorrect temperature distribution in the battery pack (primarily in the significant temperature difference) as the cause of the main problems related to the battery operation. Jing et al. [25] stressed the importance of environmental aspects in the design of vehicle cooling systems and presented the results of a research study aimed at a detailed analysis of the operating medium in terms of environmental aspects. Shen and Gao [16] focused on the properties of the cooling medium in liquid-cooled battery systems and their impact on cooling efficiency. Another team of researchers has published interesting papers on the possibility of using heat pumps to cool car batteries. Dong et al. [7] and Wang et al [22] have shown in their experimental studies that such solutions have a high application potential in motor vehicles. Chen et al. [6] confirmed their conclusions by proposing a carbon dioxide-based heat pump to cool electric vehicle batteries. However, the researchers Qiu and Shi [14], presenting their theoretical analysis results, lean toward thermoelectric cooling.

The system control method is also an important direction of BTMS development. The future in this area may be the process proposed by Miranda et al. [13]. The authors of the paper, to face the future challenge of energy management of electric vehicles, propose a fuzzy logic control (FLC) strategy. The solution is applied to perform the power split among the electric motors to improve vehicle energy efficiency and dynamic performance under real driving conditions [13].

The problem of EV energy management in real drive conditions, especially in urban area, was also analyzed by Sun et al. [20]. The presented study proposed to simulate energy consumption of electric vehicles using real-world driving cycle (RWDC) data in urban area and developed method for formulating energy optimization schemes for electrical commercial vehicles.

Considering the problems of EV batteries cooling, it is necessary to take into account the physical phenomena occurring in the system, primarily the phenomena of heat transfer and phenomena occurring during the flow of cooling agents. Observation and analysis of flow and temperature distribution is crucial for the design of flow systems, which are certainly cooling systems. Only advanced numerical tools of fluid mechanics, which enable multicriterial visualization and then flow observation, allow to design highlyeffective solutions characterized by desired parameters.

The dynamic progress in the field of technical sciences in recent decades has made it necessary to develop mathematical and numerical tools to help solve many scientific concerns (not only current but also predicted [2, 8–10, 28]) and engineering problems in the design, improvement of efficiency and performance parameters of machines and devices [28]. Computational fluid mechanics (computational fluid dynamics), is a separate area of fluid mechanics in which numerical methods are used to solve problems with the description of fluids.

Discretization and numerical solution of partial differential equations describing the behavior of the fluid make it possible to approximate the distribution of physical quantities, such as velocity, pressure, temperature, and other flow parameters.

Modern CFD programs solve: flows taking into account the variability of viscosity and compressibility, multiphase flows, flows in which chemical reactions or combustion processes occur, flows through porous structures and flows in which the medium is a Newtonian or non-Newtonian fluid. Therefore, CFD simulations are now used to solve problems in many areas of science and industry (i.e., [2, 8–10, 19, 28]).

Numerical fluid mechanics, which is a research tool enabling the modeling of the effects of flow phenomena both for the purposes of analysis and diagnostics of systems, as well as their multicriteria optimization, has been used since the 1990s as a research method for vehicle design, and in recent years in electric vehicles, in particular in solving problems of aerodynamics, energy management, and noise modeling. In Zawiślak's paper [28], a review of the research conducted so far was made and then a method for designing flow machines and systems using CFD methods on the micro and macro flow scale was proposed and described. Three years later, Ruiqing et al. [17], reviewing the applications of CFD methods in the last decade (2010-2020), conclude that it is not only a tool for the diagnosis and design of flow devices and systems, but also a tool for a better understanding of the physical phenomena occurring in the flow system, indicating numerous possibilities for the use in the latest solutions in nonconventional vehicles (including electric vehicles).

Haowen et al.'s [26] paper presents the problem of noise distribution in the electric vehicle cabin, taking into account multiple sources. The authors indicate that the use of three-dimensional CFD calculations on the basis of a two-equation k-ε turbulence model allows for very good consistency of the calculation results with the measurement results. Zhang et al. [29] applied numerical fluid mechanics to optimize the vehicle cooling system taking into account the geometry of the entire vehicle under operational conditions. The researchers' interest was focused on optimizing the radiator and the cooling drag. Due to the use of CFD methods, design recommendations for efficient vehicle cooling have been specified.

Recently, there has been a significant increase of the numericians' interest in the cooling of electric vehicle batteries. Hamidreza et al. [3], using CFD methods, performed a comparative analysis of different hybrid vehicle battery cooling techniques and methods to assess thermal management efficiency. Also Benabdelaziz et al. [4] successfully use CFD tools to test battery cooling efficiency with the developed solution in various geometric variants. The result of the execution of their work is an improvement in the operation of the cooling system based on proprietary solutions (including heatpipe [29]), understood as a reduction of the temperature gradient on the battery pack. Yuan et al. [27] identify numerical and co-numerical modeling of flows based on the latest available computational fluid mechanics tools as the right direction for the design of cooling systems for vehicle components.

2. Research subject and purpose

The object of the research were radiators with an air distribution system in the housing, intended for use in cooling a pack of lithium-ion batteries / fuel cells. The developed solution can be applied in buses with an electric drive, including those with an optional braking resistor and in vehicles with a hydrogen drive. The cooling systems have been designed using the best available technologies in terms of material selection, type and shape of the core cross-section and verified using CFD methods.

The modules were equipped with a set of fans, liquid and air temperature sensors, a wire harness adapted to communication via the CAN 2.0B bus, with a transmission speed of 250 Kbps. Dedicated control systems with proprietary software that enable autonomous operation with the lowest possible demand for electricity have been developed and manufactured. The view of the research object – the object of numerical analysis – is shown in Fig. 1.

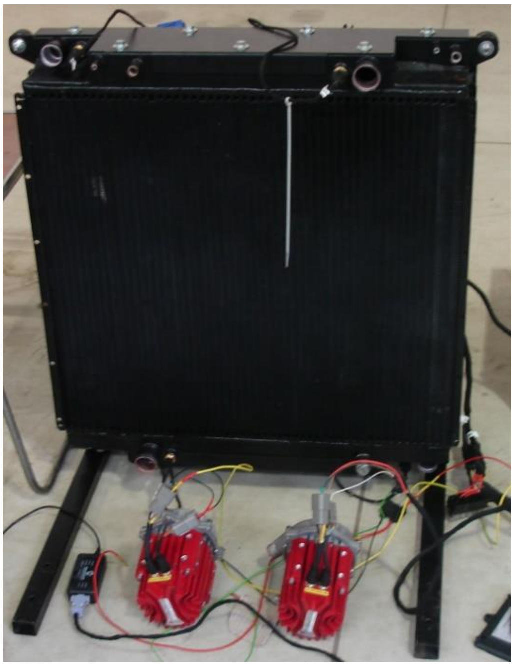

Fig. 1. The research object – radiator with an air distribution system, equipped with a set of fans, liquid and air temperature sensors, a wire harness adapted to communication via the CAN 2.0B bus, with a transmission speed of 250 Kbps

The aim of the research was numerical optimization of the device for minimizing noise emissions (improving the comfort of bus users) while maintaining the desired high efficiency of cooling the battery pack.

3. The research methods

For optimization of the electric bus radiator design in terms of noise emissions and energy consumption, the Zawiślak's method was applied [13]. The method is presented graphically in Fig. 2.

Numerical (simulation) tests were performed using the finite volume method (FVM), with the application of numerical fluid mechanics methods. ANSYS-FLUENT software was used. The flow of the cooling medium (air) was modeled using the RANS method. The two-equation k-ε turbulence model, the heat transfer model and the acoustic model were used. In order to determine the flow velocity boundary conditions, the available characteristics of the fan intended for use in the designed radiator were used. The assumed air temperature was 298 K and the assumed fluid temperature in the radiator was 373 K.

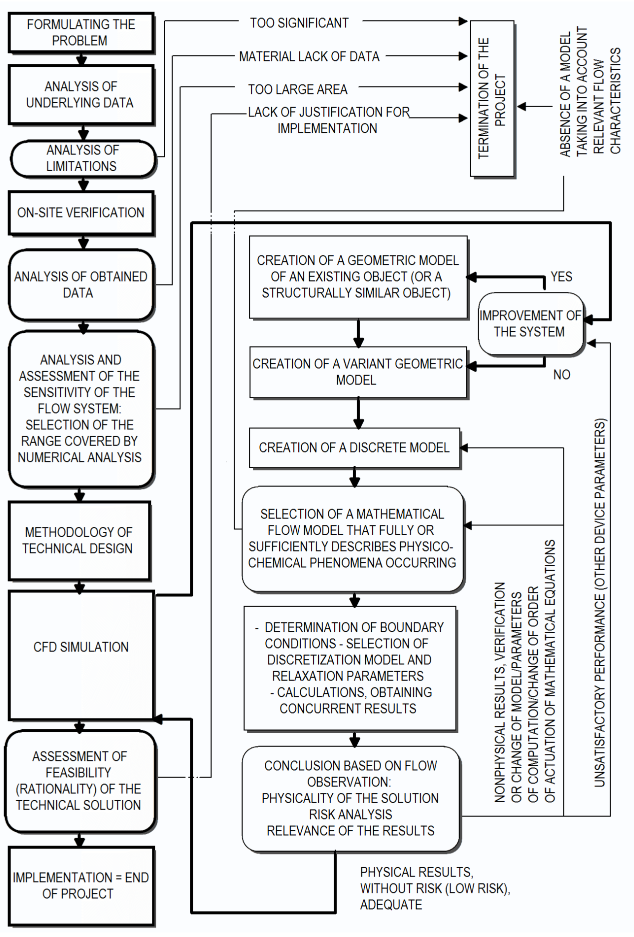

Fig. 2. Scheme of Zawiślak's method 'Design and modernisation of machines and flow systems using CFD software' [13]

Based on the results of numerical simulation, a prototype was created. In the next steps, the prototype was tested under laboratory and field conditions in order to perform a two-stage validation of the results of CFD calculations. The research methodology scheme is shown in Fig. 3.

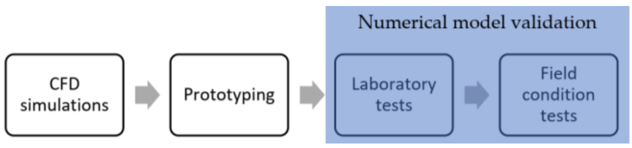

Fig. 3. Scheme of the research methodology

Measurements in laboratory conditions were carried out at the Division of Automotive Engineering of Wrocław University of Science and Technology. A test stand was built, on which the supplied system was installed, consisting of a radiator with connecting wires, a circulating pump mounted between the buffer tank and the radiator (the coolant is sucked from the buffer tank and then pumped into the radiator), a flow meter mounted on the return wire from the radiator (in front of the buffer tank). A diagram of the stand is shown in Fig. 4.

The buffer tank filled with coolant (water and glycol mixture, 5% vol. glycol), with a total volume of 65 dm^3, was equipped with three heating units with thermostatic systems with 18 kW of electric power. The system has 4 temperature sensors (PT-100); 2 sensors measuring the temperature of the coolant (coolant inlet and outlet from the radiator) and 2 sensors measuring the air temperature (air in front of the radiator and air behind the radiator at a distance of 1 m from the radiator. Electrical value measurements were made using a voltmeter, a current clump and an oscilloscope with recording software. Due to the type of control system, the current was measured by measuring the battery charging current.

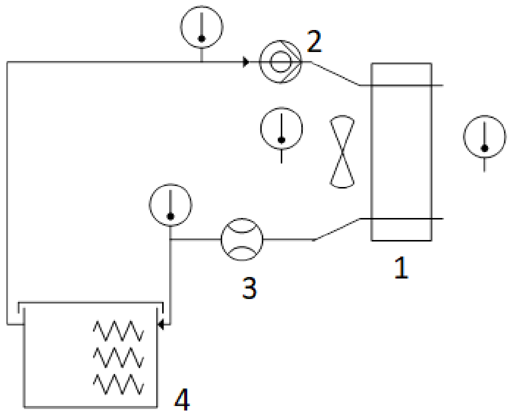

Fig. 4. A schematic diagram of the test stand: 1 – radiator, 2 – pump, 3 – flow meter, 4 – buffer tank

The characteristics of the coolant flow were determined depending on the load, expressed as a percentage of the load defined by the software. In addition, the characteristics of the current consumption of the circulation pump and cooling fans were determined, depending on the load defined as above. Additionally, oscilloscope waveforms of the current values at the device start-up were recorded. The maximum instantaneous values of current during start-up (at a nominal voltage of 27.1 V) did not exceed the nominal values of the devices.

Measurements in real (field) conditions were made at the Military Institute of Armoured and Automotive Technology on specially constructed measuring stands, according to a specially developed method (test certificate 41/LPE/2020) using a digital multimeter with instrumentation and a set of power batteries (device power) and a type 2250 sound level meter (noise level).

4. Results

Before starting the CFD simulations, numerical meshes were established based on the geometry of the bus and the radiator. The numerical mesh of the bus geometry was 1.2 million cells and 900,000 cells for the radiator. The tetra cells were applied in accordance with modeling principles [13] as suitable for modeling complex geometries.

Figure 5 shows a discrete model (a numerical mesh based on tetra cells) of the vehicle in which the analyzed radiator is planned to be installed.

A simulation of airflow around the vehicle was performed to determine whether there are no adverse flow phenomena at the cooling system attachment point (shown in Fig. 5).

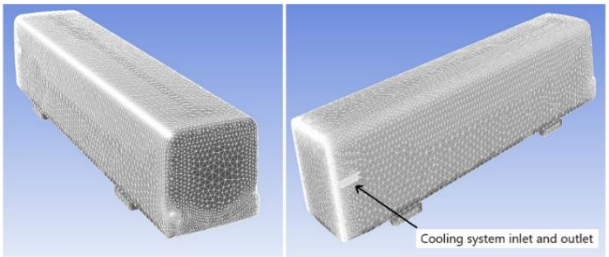

Fig. 5. A discrete model of an electric vehicle (bus)

Figure 6 presents the results of calculations showing the behavior of the streamlines during the flow under the given speed conditions (90 km/h).

Streamlines – radiator inlet grilles and streamlines with operating fans are shown in Fig. 7.

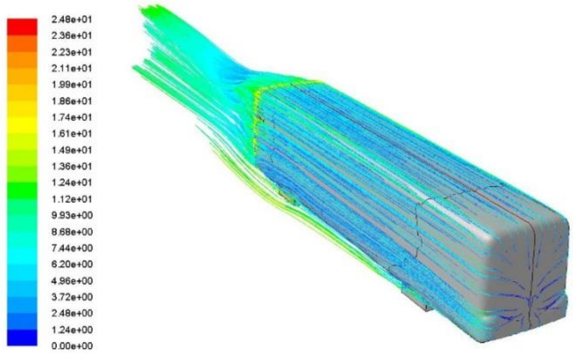

Fig. 6. Streamlines based on velocity [m/s] – view of the bus from the front of the vehicle on the side of radiator mounting

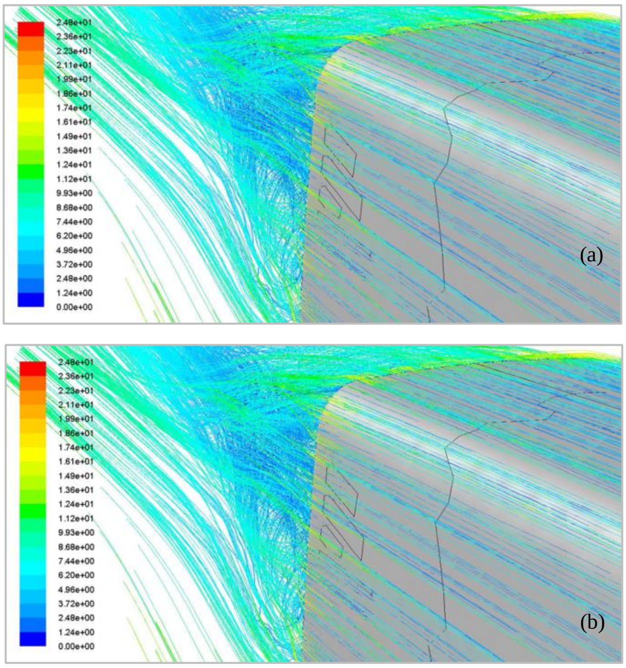

Fig. 7. Streamlines based on velocity [m/s] – radiator inlet grilles and streamlines with operating fans: view of the inlet grilles: a) when the vehicle is moving without the cooling system on, b) when the vehicle is moving with the cooling system on

Despite the correct air flow around the vehicle (Fig. 4), it was found that when the radiator fans were switched on, the flow into and out of the radiator box has swirl zones and dead-zones of the flow. To improve the flow condition (air inlet and outlet from the system), an analysis of the flow inside the radiator attachment point with operating fans (inside the radiator housing) was performed. The results of the calculation are presented in Fig. 8.

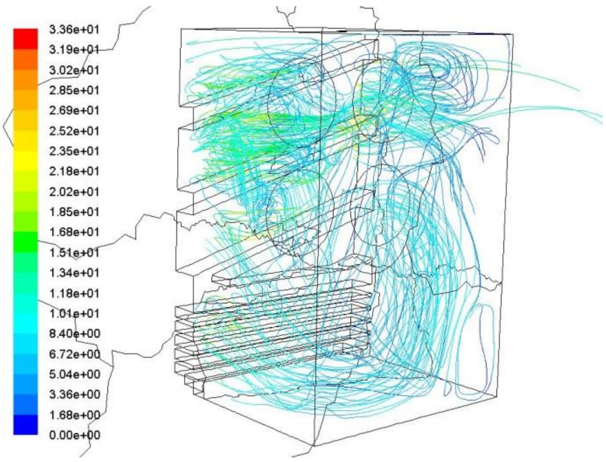

Fig. 8. Streamlines based on velocity [m/s] inside the radiator box

Observation of the streamlines indicates the need for precise direction the stream flow through the use of guide vanes.

The simulation results showed dead-fields of the flow, i.e. a decrease in the efficiency of the radiators and noise levels above 72 dB. Additionally, the position of the fans was found to be correct (no stream detachment and no flow swirl). However, flow dead-zones were observed during the operation of the fans. It was decided to redesign the fan-housing box. Four geometric models were selected for analysis (Fig. 9). They were modeled at the operating point corresponding to 50% of the fan load in the following geometry variants: (a) without a guide vane, (b) with one guide vane, (c) with two guide vanes, (d) with stream separation.

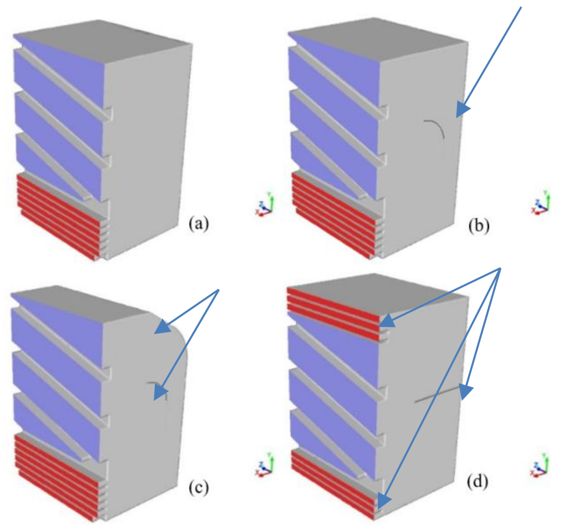

Fig. 9. Geometric models: (a) without guide vanes, (b) with one stream guide vane, (c) with two stream guide vanes, (d) with stream separation. The changes in geometry are indicated by arrows

Figure 10 shows the simulation results in the form of a determined distribution of acoustic pressure fields.

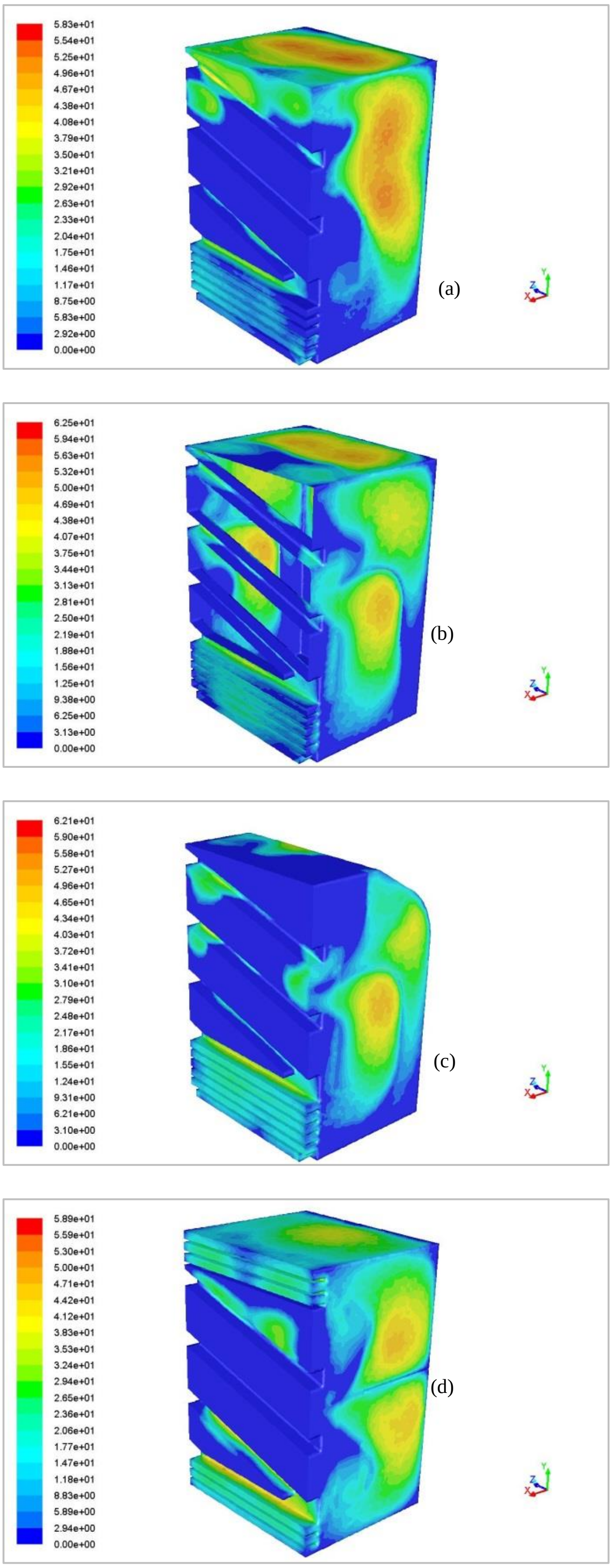

Fig. 10. Acoustic pressure fields [dB]: (a) without guide vanes, (b) with one stream guide vane, (c) with two stream guide vanes, (d) with stream separation

Velocity vector fields are shown in Fig. 11, while inlet and outlet velocity vector fields are presented in Fig. 12.

As a result of the simulation, it was found that the best solution in terms of noise emission as well as the flow phenomena that occur was obtained for variant d) – the variant ant d), no disadvantageous flow phenomena were observed (as opposed to other variants) like presence of dead-zones in the area of the inlet and outlet of the cooling medium and

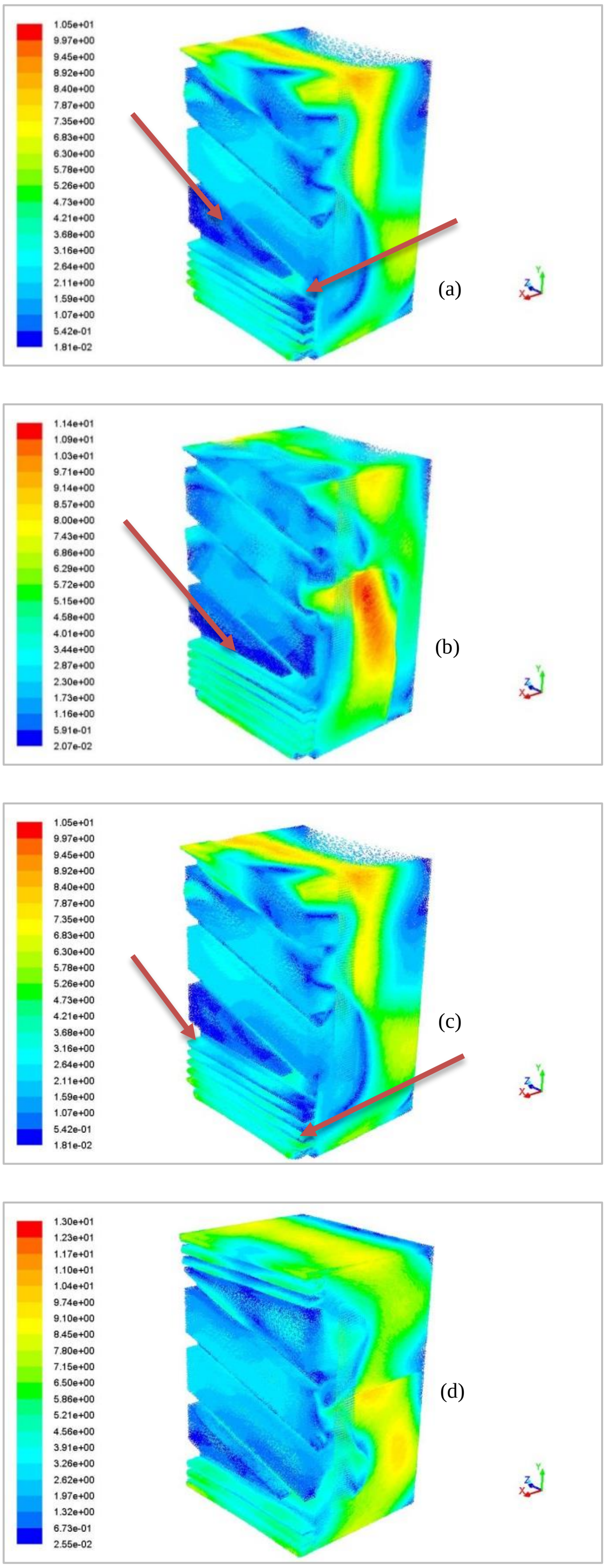

Fig.11. Velocity vector fields [m/s] (a) without guide vanes, (b) with one stream guide vane, (c) with two stream guide vanes, (d) with stream separation. The dead-zones and adverse flow events are indicated by red arrows

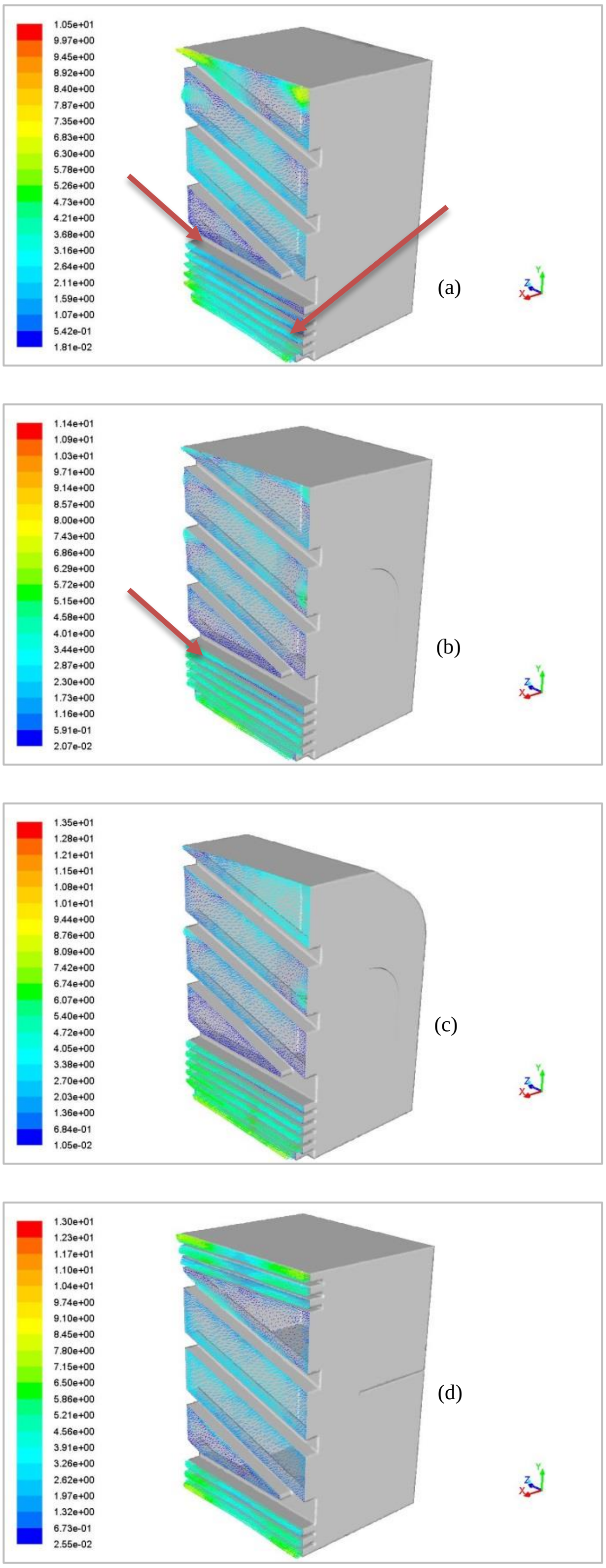

Fig. 12. Inlet and outlet velocity vector fields [m/s] (a) without guide vanes, (b) with one stream guide vane, (c) with two stream guide vanes, (d) with stream separation. The dead-zones and adverse flow events are indicated by red arrows

with the stream separation. The maximum values of the noise level for this variant were estimated at 57 dB. In vari return flow. The separation of the air stream improved the distribution of the velocity field, the pressure field and the acoustic field. A more even distribution of the velocity fields results in improved cooling efficiency while maintaining the same surface area of the cooling elements. Obtaining an even flow through the radiators also allows for the best cooling performance (energy efficiency).

The geometry of the system according to the d) variant is shown in Fig. 13.

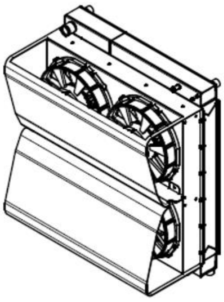

Fig. 13. Selected radiator geometry

In the next stage, empirical verification of the numerical model was conducted. A prototype device was made according to the geometric model selected as a result of simulation tests. The experimental tests were conducted under field conditions at the Military Institute of Armoured and Automotive Technology. Figure 14 shows the view of the test stand during the study implementation.

Comparative tests of the base solution (Fig. 1) and the solution resulting from the numerical tests (Fig. 13) were performed. The results of noise level measurements – comparison of the base and prototype versions (developed on the basis of numerical analysis) are shown in Fig. 15.

The accuracy of the calculation model was found to be very good – under simulation conditions, the maximum noise level was estimated at 57 dB, under measurement conditions for the same operating point (50% of fan power) at 59 dB.

The measurements were then carried out to determine the prototype power requirement in laboratory and field conditions according to the measurement methods described in Chapter 3.

Figure 16 shows the results of laboratory tests, while Fig. 17 shows the results of field tests.

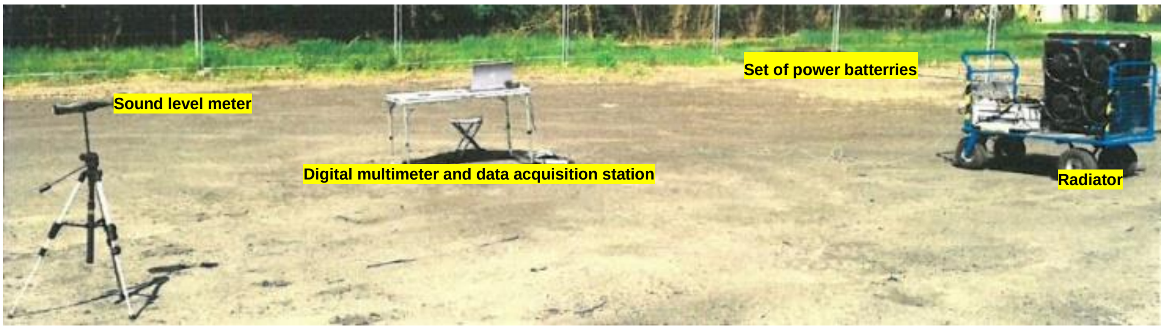

Fig. 14. The view of the test stand during the performance of acoustic pressure tests of the radiator in the field conditions at the Military Institute of Armoured and Automotive Technology (test certificate 41/LPE/2020)

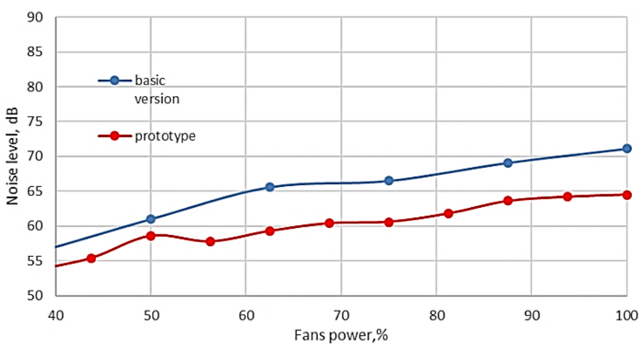

Fig. 15. Results of noise level measurements – comparison of the base and prototype versions (developed on the basis of numerical analysis)

It was found that the determined characteristics were convergent and that there were no changes in the electrical power requirement of the device after the optimization performed.

Based on the results of the power measurement in field conditions tests the model revealing the dependency of power requirement and fan power was developed. Figure 17 shows the curve fitted to the observations. The RMSE of the model is 219.9 and R^2 = 0.9958. The power of the model is determined by comparing the residuals to the Gaussian distribution.

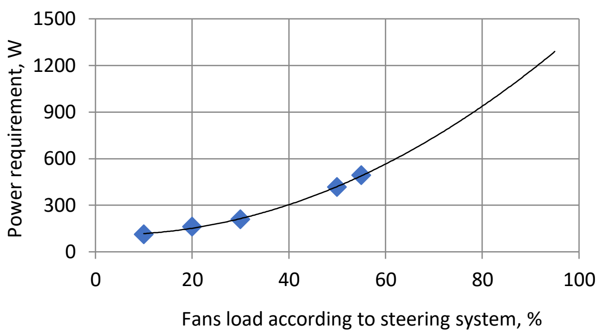

Fig. 16. Cooling fan power consumption depending on the % of the load defined by the control system in laboratory conditions

The verification was done by hypothesis testing and with the aid of the Shapiro-Wilk Kolmogorov Smirnov test. In this case the p = 0.3504. Therefore, there is no reason to

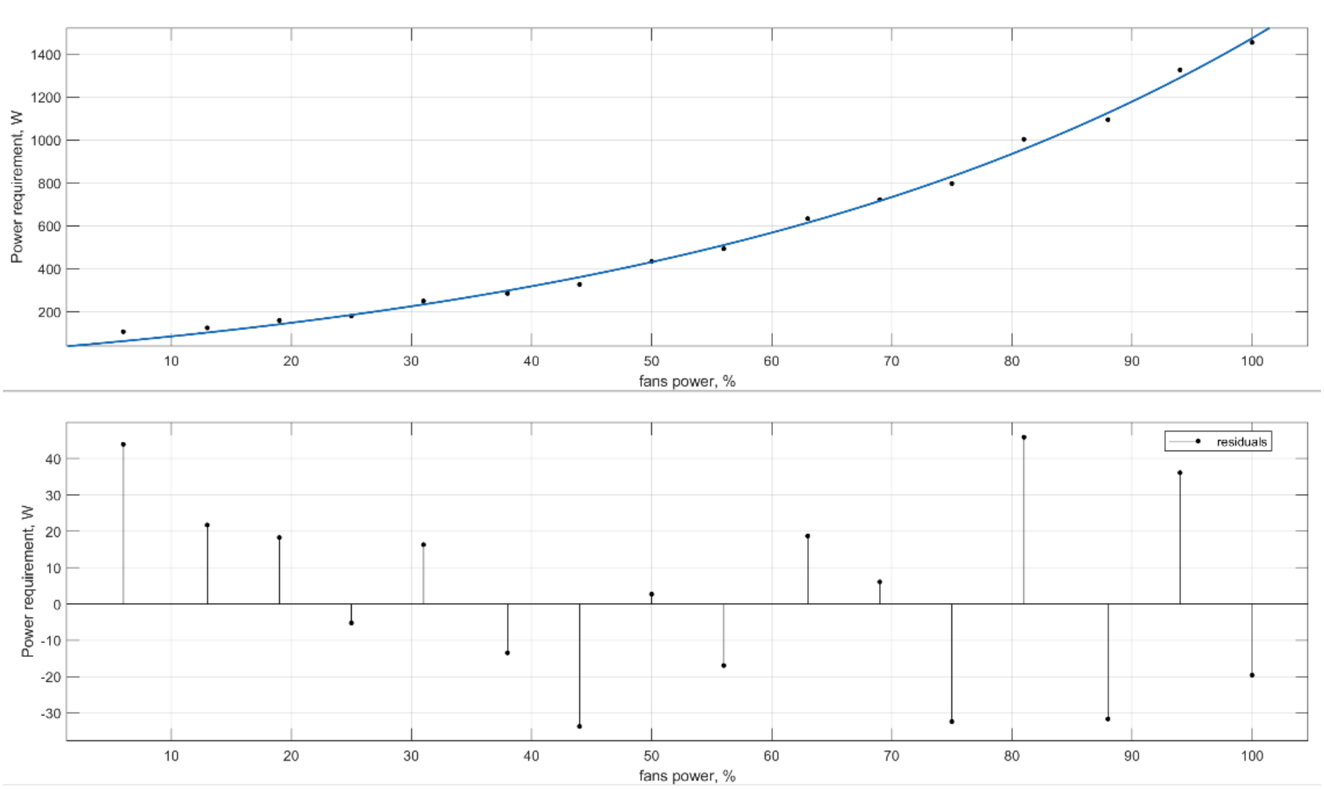

Fig. 17. Observed parameters of power requirement, W, with respect to fan load, %, in field conditions

reject the null hypothesis that constitutes the Gaussian distribution of the residuals. The relatively high value of the p coefficient proves that the model is accurate and reliable.

Based on the results it was found that the determined characteristics were convergent and that there were no changes in the electric power requirement of the device after the performed optimization

5. Conclusions

1. The application of the CFD RANS method of the two-equation k-ε turbulence model with the heat transfer and the acoustic model as a tool to optimize the vehicle's cooling system in terms of noise emission is an adequate instrument for the intended purpose. The model was validated in two stages under laboratory and field conditions. The consistency of the simulation results and test results of the real object was demonstrated, i.e., 57 dB as a result of the simulation and 59 dB on the basis of field tests.
2. As a result of the work, the design of the cooler for the electric bus battery pack was optimized. Based on the observation and flow analysis, it was decided to make a geometrical change involving the separation of the cooling medium stream, which improved the distribution of the velocity field, the pressure field and the acoustic field. During the numerical analysis of the selected geometry variant, no adverse flow phenomena were observed, such as the presence of dead-zones in the area of the inlet and outlet of the cooling medium and return flow.
3. The separation of the air stream flowing through the cooler resulted in the reduction of the maximum velocity values, which had a direct impact on the reduction of the flow resistance.
4. Reducing the pressure level and obtaining an even flow through the radiators allowed for a higher cooling efficiency while maintaining a comparable level of electric power.
5. The presented results indicate a new approach to the method of shaping internal elements of electric vehicle coolers for improving energy efficiency, which translates directly into economic indicators (also resulting from battery life) and reduces noise emissions.
6. The good convergence of the model with the results of the experiments points to the possibility of developing generalized guidelines for designers of electric vehicles, which, however, requires further research based on CFD analysis with the proposed method of calculating other design solutions for different battery powers and their location in the vehicle.
7. The results of the research indicate the necessity to change the priorities in the process of designing cooling systems in vehicles, taking into account the flow phenomena in the first place (currently these are mainly visual aspects).

Acknowledgements

The research was conducted as part of a project co-financed by the National Centre for Research and Development entitled: "Innovative generation of ultra-light cooling modules with minimized noise emission, electricity demand, through the implemented new system of ongoing monitoring of operating parameters, intended for new electric vehicles, including those powered by hydrogen cells", project no: POIR.01.01.01-00-0147/17.

Bibliography

[1] AKINLABI, H.A.A., SOLYALI, D. Configuration, design, and optimization of air-cooled battery thermal management system for electric vehicles: A review. *Renewable and Sustainable Energy Reviews.* 2020, **125**, 109815. https://doi.org/10.1016/j.rser.2020.109815

[2] BAN. L., DUB. W., JIN. T. et al. A roughness parameter considering joint material properties and peak shear strength model for rock joints. *International Journal of Mining Science and Technology*. 2021, **31**, 413-420. https://doi.org/10.1016/j.ijmst.2021.03.007

[3] BEHIA, H., KARIMIA, D., BEHIC, M. et al. A new concept of thermal management system in Li-ion battery using air cooling and heat pipe for electric vehicles. *Applied Thermal Engineering*. 2020, **174**, 115280. https://doi.org/10.1016/j.applthermaleng.2020.115280

[4] BENABDELAZIZ, K., LEBROUHI, B., MAFTAH, A. et al. Novel external cooling solution for electric vehicle battery pack. *Energy Reports*. 2020, **6**, 262-272. https://doi.org/10.1016/j.egyr.2019.10.043

[5] BIAŁY, M., PIETRYKOWSKI, K., TULWIN, T. et al. CFD numerical simulation of the indirect cooling system of an internal combustion engine. *Combustion Engines*. 2017, **170**(3), 8-18. https://doi.org/10.19206/CE-2017-302

[6] CHEN, Y., ZOU, H., DONG, J. et al. Experimental investigation on the heating performance of a CO_2 heat pump system with intermediate cooling for electric vehicles. *Applied Thermal Engineering*. 2021, **182**, 116039. https://doi.org/10.1016/j.applthermaleng.2020.116039

[7] DONG, J., WANG, Y., SHIWEI, J. et al. Experimental study of R744 heat pump system for electric vehicle application. *Applied Thermal Engineering*. 2020, **183**(1), 116191. https://doi.org/10.1016/j.applthermaleng.2020.116191

[8] HU, J., XU, G., SHIA, Y. et al. A numerical simulation investigation of the influence of rotor wake on sediment particles by computational fluid dynamics coupling discrete element method. *Aerospace Science and Technology*. 2020, **105**, 106046. https://doi.org/10.1016/j.ast.2020.106046

[9] ISHIHARA, T., QIAN, G.-W., QI, Y.-H. Numerical study of turbulent flow fields in urban areas using modified k-ε model and large eddy simulation. *Journal of Wind Engineering & Industrial Aerodynamics.* 2020, **206**, 104333. https://doi.org/10.1016/j.jweia.2020.104333

[10] JOUVET, G., HUSS, M., BLATTER, H. et al. Numerical simulation of Rhonegletscher from 1874 to 2100. *Journal of Computational Physics.* 2009, **228**, 6426-6439. https://doi.org/10.1016/j.jcp.2009.05.033

[11] KIM, J., OH, J., LEE, H. Review on battery thermal management system for electric vehicles. *Applied Thermal Engineering*. 2019. **149**, 192-212. https://doi.org/10.1016/j.applthermaleng.2018.12.020

[12] LU, M., ZHANG, X., JI, J. et al. Research progress on power battery cooling technology for electric vehicles. *Journal of Energy Storage*. 2020, **27**, 101155. https://doi.org/10.1016/j.est.2019.101155

[13] MIRANDA, M.H.R., SILVA, F.L., LOURENÇO, M.A.M. et al. Electric vehicle powertrain and fuzzy controller optimization using a planar dynamics simulation based on a real-world driving cycle. *Energy*. 2022, **238**, 121979. https://doi.org/10.1016/j.energy.2021.121979

[14] QIU, C., SHI, W. Comprehensive modeling for optimized design of a thermoelectric cooler with non-constant cross-section: Theoretical considerations. *Applied Thermal Engineering.* 2020, **176**, 115384. https://doi.org/10.1016/j.applthermaleng.2020.115384

[15] SHASHANK, A. Selection of thermal management system for modular battery packs of electric vehicles: A review of existing and emerging technologies. *Journal of Power Sources.* 2018, **400**, 621-640. https://doi.org/10.1016/j.jpowsour.2018.08.020

[16] SHEN, M., GAO, G. Structure design and effect analysis on refrigerant cooling enhancement of battery thermal management system for electric vehicles. *Journal of Energy Storage.* 2020, **32**, 101940. https://doi.org/10.1016/j.est.2020.101940

[17] SHEN, R., JIAO, Z., PARKER, T. et al. Recent application of computational fluid dynamics (CFD) in process safety and loss prevention: A review. *Journal of Loss Prevention in the Process Industries*. 2020, **67**, 104252. https://doi.org/10.1016/j.jlp.2020.104252

[18] SROKA, Z.J., SADLAK, Z. Thermal activation of the combustion chamber of a reciprocating internal combustion engine. *Journal of Thermal Science*. 2018, **27**(5), 449-455. https://doi.org/10.1007/s11630-018-1039-7

[19] SUN, D., SHI, X., ZHANG, Y. et al. Spatiotemporal distribution of traffic emission based on wind tunnel experiment and computational fluid dynamics (CFD) simulation. *Journal of Cleaner Production*. 2021, **282**, 124495. https://doi.org/10.1016/j.jclepro.2020.124495

[20] SUN, D., ZHENG, Y., DUAN, R. Energy consumption simulation and economic benefit analysis for urban electric commercial-vehicles. *Transportation Research Part D: Transport and Environment*. 2021, **101**, 103083. https://doi.org/10.1016/j.trd.2021.103083

[21] UÇAR, S., UÇAR, E., ÖZDEMIR, N. et al. Mathematical analysis and numerical simulation for a smoking model with Atangana–Baleanu derivative. *Chaos, Solitons and Fractals*. 2019, **118**, 300-306. https://doi.org/10.1016/j.chaos.2018.12.003

[22] WANG, Y., DONG, J., SHIWEI, J. et al. Experimental Comparison of R744 and R134a heat pump systems for electric vehicle application. *International, Journal of Refrigeration*. 2020, **121**, 10-22. https://doi.org/10.1016/j.ijrefrig.2020.10.026

[23] WORSZTYNOWICZ, B. Experimental verification of design calculations of the internal combustion engine cooling system. *Combustion Engines.* 2019, **179**(4), 132-135. https://doi.org/10.19206/CE-2019-421

[24] WORSZTYNOWICZ, B. Influence of the method of implementing the forced air flow through the cooling system on the temperature of the coolant in heavy-duty engines. *Combustion Engines.* 2017, **171**(4), 51-55. https://doi.org/10.19206/CE-2017-409

[25] WU, J., ZHOU, G., WANG, M. A comprehensive assessment of refrigerants for cabin heating and cooling on electric vehicles. *Applied Thermal Engineering*. 2020, **174**, 115258. https://doi.org/10.1016/j.applthermaleng.2020.115258

[26] YUAN, H., SU, H., WANG, L. et al. Numerical analysis on airflow and thermal field in quiet power vehicle compartment. *Procedia Engineering*. 2017, **174**, 571-578. https://doi.org/10.1016/j.proeng.2017.01.189

[27] YUAN, R., FLETCHER, T., AHMEDOV, A. et al. Modelling and co-simulation of hybrid vehicles: a thermal management perspective. *Applied Thermal Engineering*. 2020, **180**, 115883. https://doi.org/10.1016/j.applthermaleng.2020.115883

[28] ZAWIŚLAK, M. Method of designing and modernising machines and flow systems using Computational Fluid Mechanics. *Poznan Publishing House*, Poznan 2017.

[29] ZHANG, C., UDDIN, M., ROBINSON, A.C. et al. Full vehicle CFD investigations on the influence of front-end configuration on radiator performance and cooling drag. *Applied Thermal Engineering*. 2018, **130**, 1328-1340. https://doi.org/10.1016/j.applthermaleng.2017.11.086

Prof. Anna Janicka, DSc., DEng. – Mechanical Faculty, Wroclaw University of Science and Technology.
e-mail: *anna.janicka@pwr.edu.pl*

Artur Głogoza – OE INDUSTRY, Poland.
e-mail: *a.glogoza@oeindustry.com*

Maciej Zawiślak, DSc., DEng. – Mechanical Faculty, Wroclaw University of Science and Technology.
e-mail: *macej.zawislak@pwr.edu.pl*

Radoslaw Włostowski, MEng. – Mechanical Faculty, Wroclaw University of Science Technology.
e-mail: *radoslw.wlostowski@pwr.edu.pl*