27th ABCM International Congress of Mechanical Engineering
December 04-08, 2023, Florianópolis-SC, Brazil

COB-2023-0494
ANALYSIS VIA CFD OF THE BATTERY INTERCONNECTION PIPELINE CIRCUIT TO THE ELECTRIC BUS THERMAL MANAGEMENT SYSTEM

André Yudi Lima Okamoto[1]
Danilo Albuquerque Ribeiro[2]
Leandro Oliveira Salviano[3]
Aluisio Viais Pantaleão[4]
São Paulo State University, School of Engineering, Department of Mechanical Engineering, 15385-00, Ilha Solteira, SP, Brazil
[1]andre.yudi@unesp.br
[2]danilo.albuquerque@unesp.br
[3]leandro.salviano@unesp.br
[4]aluisio.pantaleao@unesp.br

Abstract. *The use of numerical methods for predicting phenomena associated with a flow plays an important role in the development of new technologies and improvement of previously existing technologies in the area of Fluid Mechanics. In this way, the study of calculation methodology based on modern CFD (Computational Fluid Dynamics) corresponds to a great step towards the understanding of phenomena physics related to the flow, and can be used in a series of applications, such as ship hydrodynamics, aircraft aerodynamics, biomedical engineering, among others. The purpose of this work is to design an optimized duct circuit that interconnects the batteries to the thermal management system of an electric public urban transport vehicle (BEV – Battery Electric Vehicle), using free codes (open source software), and contribute to increasing the vehicle's autonomy. One of the most important features of this cooling system is suitability for the operating climate, as that a change of 0.4 °C in the system can cause considerable loss of autonomy. For this study, different types of proposed insulator were simulated individually subject to a constant temperature in the external surface, with the possibility of multilayer if necessary. Using the OpenFOAM software, steady-state numerical simulations involving heat transfer were performed for the different types of insulation in order to obtain the maximum thermal efficiency. Numerical results obtained using the RAS k-ω SST turbulent model were expected to be close to the flow topology reported by experimental literature. It is proposed, then, to validate the numerical techniques developed here from databases of data from vehicle tests, specifically applied to the pipelines of the thermal management of the battery pack. The results allowed to determine the optimal cooling system among series of insulation, that provides the best thermal efficiency, contributing to autonomy of the vehicle. In addition, the robust and customized metodology employed allows extrapolation to other thermal vehicle systems.*

Keywords: *CFD, OpenFOAM, interconnection ducts, thermal management, suitability to the operating climate*

1. INTRODUCTION

The worldwide fleet electrification is currently in rapidly progress. Amidst the demand for emission reduction, big companies plan to electrify 100 % of their fleet by the 2030s. Brazil follows this trend. According to 2022 data, electric car sales increased by 78 % in the first quarter. In this way, many tools are under development to improve the existent electric vehicles or to enable these technologies to operate properly.

The development of electric vehicles is a challenge for companies installed in Brazil, since their matrices are mostly localized in Europe, Asia or North America. This vehicles are projected for conditions different from those found in Brazil. The main concern is the climate conditions, which are much more aggressive in Latin America. It can be seen by comparing the temperatures along the past year in Sevilla, where the bus was originally developed, and Ilha Solteira, where the bus will be tested (Fig. 1).

The battery performance and, consequently, the vehicle autonomy are sensible to coolant temperature. For the application in large vehicles, like collective transport, it is common for pack of batteries to be far from the thermal management system. The interconnection between the components is made through pipes, that can be exposed to thermal source as solar radiation, hot air recirculation, forced convection by vehicle motion, besides the thermal dissipation of the batteries.

Tete *et al.* (2021) explored various factors and techniques to enhance the thermal management system of electric vehicle battery packs. These factors include battery cell layout, heat pipe cooling, interaction between the coolant and battery surface, among other methods. In their research, Zhao *et al.* (2021) have suggested that by employing computer numerical simulations and advanced experiments, novel concepts for battery packs can be developed. This highlights the significance of Computational Fluid Dynamics (CFD) simulations in advancing new technologies and studying fluid

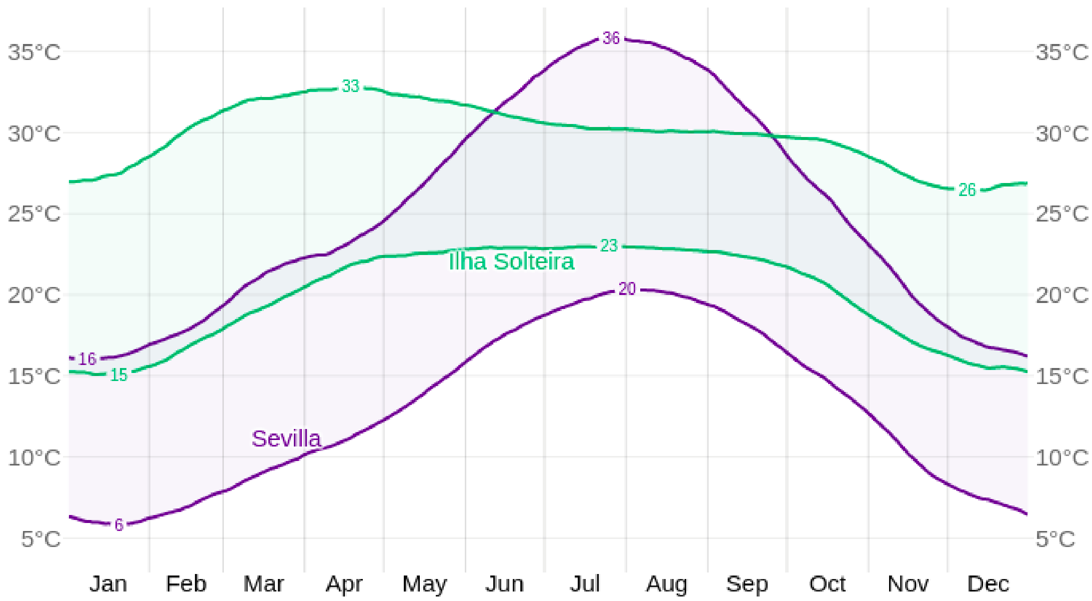

Figure 1. Maximum and minimum temperature comparison between Sevilla and Ilha Solteira (Spark (2022))

dynamics and heat transfer.

Karimi and Li (2013) conducted numerical simulations and found that a cooling strategy based on forced convection could facilitate a uniform temperature distribution within the battery pack. Chen *et al.* (2018) utilized CFD techniques to optimize a cooling channel in a battery system, aiming to enhance the cooling rate. Chen *et al.* (2019) concluded that CFD allows to find the ideal solution for thermal management system regardless of design and operating parameters. This emphasizes the importance of CFD in the field.

Alternatively, Mondal *et al.* (2018) proposed another approach by introducing vortex generators into the cooling channel of the thermal management system. They discovered that this modification intensifies the heat transfer rate, resulting in a reduction of the system's maximum temperature.

The present work aims to develop a robust methodology to optimize the interconnection pipe circuit, that connects the battery pack to the thermal management system, which is originally developed for the climate of Europe. The suitability to higher temperatures of Brazil is a key feature that impacts the efficiency of this system. Therefore, this projects could contribute to the insertion of electric vehicles in national territory. Furthermore, this methodology allows extrapolation to another projects that employ CFD techniques and optimization.

2. MATERIALS AND METHODS

2.1 Computational resources

This research used a workstation provided by Computational Support Center of São Paulo State University, campus of Ilha Solteira. Using 32 CPU's, the entire simulations (until convergence) took a simulation time varying from less than 5 minutes (coarser meshes) to 4 hours (finer meshes).

2.2 Geometry and mesh generation

The pipe is made of stainless steel EN 1.4301, whose characteristics are: thermal conductivity of 16.2 W/mK and thickness of 0.0015 m. The initial geometry is preset by the bus production company and can be seen in Tab. 1 and Fig. 2. But this geometry can be changed by the optimization if this results in better flow conditions. The main limitation is that this pipe has to fit in the bus configuration.

The proposed insulation layer was suggested to consist of a multi-layer configuration using two commercially accessible insulation types known as thin insulation and thick insulation. The information about the materials, their physical characteristics, and cost per meter can be seen in Tab. 2. In assessing the optimal configuration, two key factors were taken into consideration: the ability of the configuration to minimize the temperature differential between the inlet and outlet, and the overall cost of the arrangement.

The results of this work were obtained through the implementation of the finite volume method, whose bases are described in Versteeg and Malalasekera (2007) and Maliska (2004). The software chosen to implement this method was OpenFOAM, an open-source software and widely validated in the market. The mesh was generated using the *blockMesh* tool, a native structured mesh (only hexahedral cells) generator. A structured mesh provides briefly three advantages over

Table 1. Dimensions of the geometry

Part of the geometry	Length m
Inlet length	0.0758
First intermediary length	0.233
Second intermediary length	5.18
Outlet length	0.257
Pipe radius	0.01
First curve radius	0.04
Second curve radius	0.08
Third curve radius	0.04

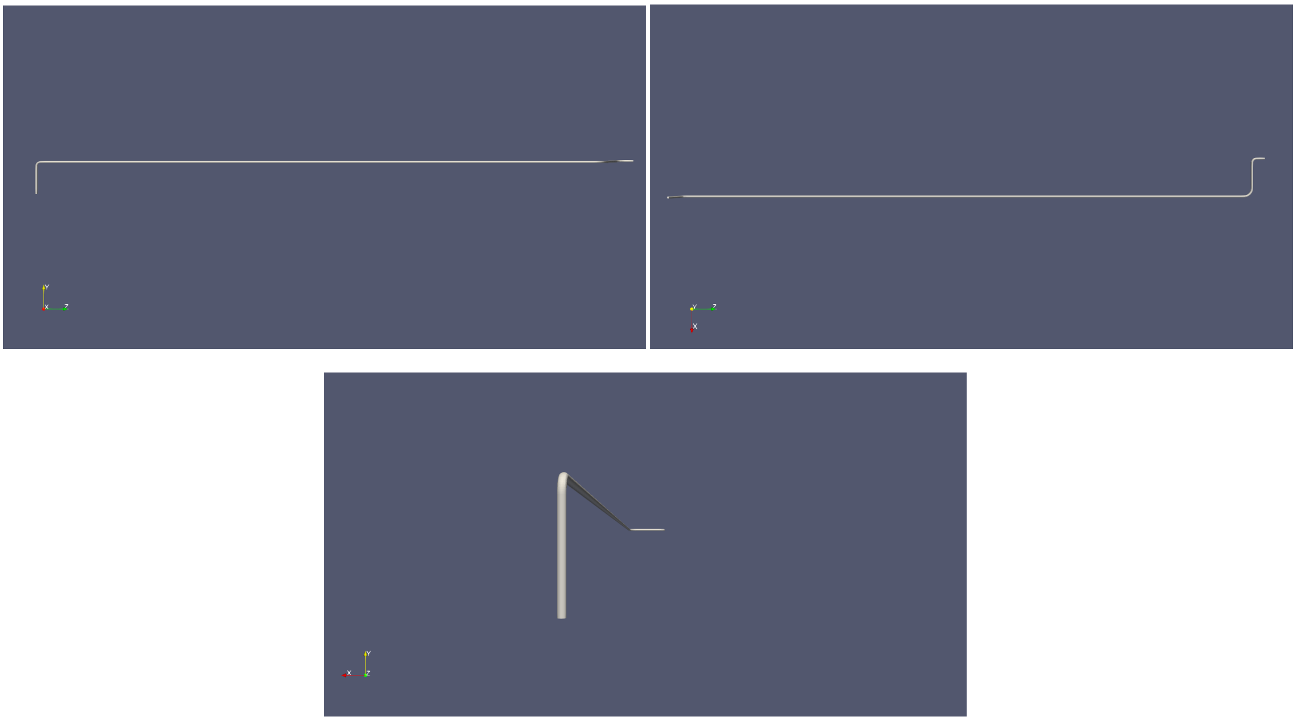

Figure 2. Geometry view direction set to +X, +Y and +Z respectively

Table 2. Proposed insulation

Property	Thin insulation	Thick insulation
Material	Armaflex tape (elastomeric insulation)	Expanded polyethylene foam
Thickness [mm]	3	10
Thermal conductivity [W/mK]	0.037	0.036
Cost per meter [R$]	9.26	4.00

a non-structured mesh: greater resolution at the wall, efficient volume fill and minimization of the numerical diffusion. The resolution at the wall is important due to the fact that the gradients are steeper in this region. The use of a non-structured mesh results in a larger number of cells, increasing the dimensions of the matrices, taking a lot more time to solve. Despite presenting a larger number of cells, a non-structured mesh doesn't provide a better space filling and resolution, since hexahedral cell fill the volume more efficiently than other polyhedral cells with same or similar level of resolution. Numerical diffusion arises from the numerical discretisation of the advection term and reduces the accuracy of the solution. Numerical diffusion can be reduced by refining the mesh or higher-order discretisation. We want to align the cells with the flow direction, but tetrahedral cells and pyramids are never aligned with the flow direction. Hexahedral cells can be aligned with the flow direction, if the mesh is designed carefully.

The O-grid approach, which is used in this paper, addresses many of the problems cited above. This method allows to add some resolution normal to the wall to catch high gradients in this region. Furthermore, the O-grid is composed by only hexahedral cells, reducing numerical diffusion. The curved O-grid provides a improvement in relation to the standard

O-grid (with square central part), it reduces the skewness of the cells in transition to the core part, being the best approach for pipelines.

The mesh described above is generated by an algorithm into a shell script. In this way, the mesh is automatically generated by inserting the required dimensions. Then, the script performs the necessary calculations and substitutes the necessary values for mesh generation within the *blockMeshDict* file. This algorithm, in addition to facilitating mesh generation, also facilitates coupling with the optimizer, reducing system complexity and simulation time.

The mesh convergence analysis was based on procedure showed in Celik *et al.* (2008), which is a implementation of the Richardson Extrapolation (RE), originally described in Richardson (1911) and Richardson and Gaunt (1927). This method is recommended by the literature to estimate the discretization errors.

Table 3. Estimation of uncertainty due to discretization

Volume of mesh 1 V_1	1
Volume of mesh 2 V_2	1
Volume of mesh 3 V_3	1
Number of elements in mesh 1 N_1	3700000
Number of elements in mesh 2 N_2	1561500
Number of elements in mesh 3 N_3	462500
Refinement factor r_{21}	1.33
Refinement factor r_{32}	1.50
Reference property in mesh 1 ϕ_1	315.55
Reference property in mesh 2 ϕ_2	317.19
Reference property in mesh 3 ϕ_3	318.72
Coefficient s	1
Coefficient p	1.86
Convergence index GCI_{21}	0.9%
Convergence index GCI_{32}	-0.54%

Based on the results shown in Tab. 3, the coarse mesh (first mesh in Fig. 3) was adopted in this project, since the discretization error associated with this mesh is 0.54 %. This mesh provides the best computational efficiency (shorter simulation time) and results with a good accuracy. Furthermore, the choice of this mesh will save much time in the coupling with the optimization, providing a greater efficiency to the entire process.

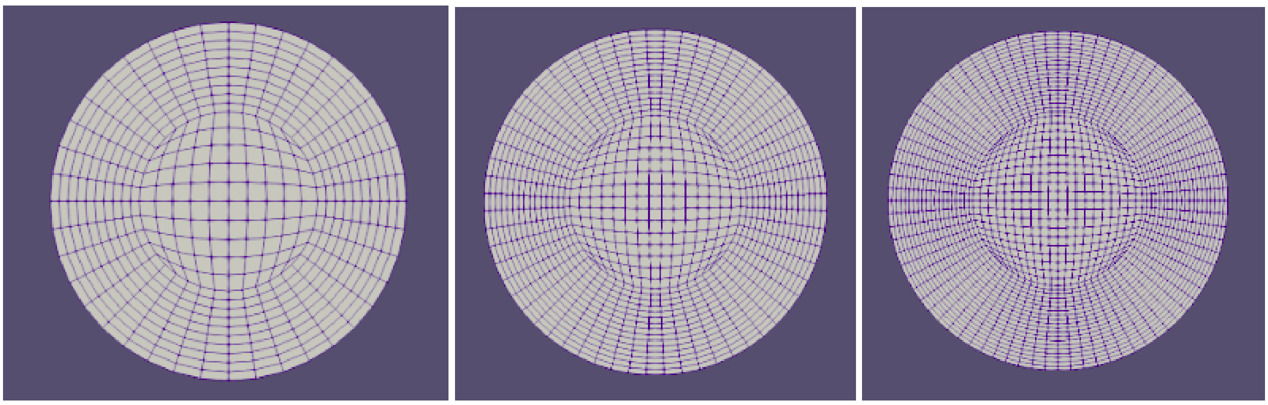

Figure 3. Meshes coarse, medium and fine

2.3 Turbulence model

The turbulence model used was RAS k-ω SST developed by Menter (1994). This turbulence model was chosen because of its robustness, since its formulation combines the best of two worlds. The k-ω (Wilcox (1988)) turbulence model is used near the walls, since its formulation provides greater precision in the viscous sub-layer. The SST formulation switches to k-ϵ (Jones and Launder (1972)) model in the free-stream, avoiding the problem of large adverse pressure gradients associated with this turbulence model.

2.4 Flow conditions and numerical solver

The conditions used are: imposed velocity at inlet, imposed pressure at outlet and imposed temperature at the wall of the pipe, being calculated at the other points. The inlet velocity can be given directly or indirectly by mass flow rate or, in this case, volumetric flow rate. The volumetric flow rate can vary depending on the cooling demand and its value is given by test data, represented by data reduction in Fig. 4. At the walls of the pipe, was adopted the no slip condition (a fluid in contact with a solid surface does not slide over it). The pressure adopted at outlet is a reference pressure. Thus, the difference between the pressure adopted at the outlet and the pressure at a given point in the duct gives the pressure drop. For the temperature, a fictitious temperature imposition on the surface of the duct was used. This temperature represents all conditions external to the pipeline, such as radiation and natural convection. To achieve this objective, this temperature was calibrated by the optimization (final part in Fig. 4) using the temperatures at inlet and outlet from the tests given by data reduction (statistical part in Fig. 4).

The fluid of work is a mixture 50/50 of water and ethylene glycol. The physical and thermal properties of the fluid are acquired using a Python script that gets these properties from the Glysantin catalog SE (2022) for each specified temperature through a csv file.

The solver used was *buoyantSimpleFoam*. It's a steady-state solver for buoyant, turbulent flow of compressible fluids. It means that this solver includes the continuity equation (conservation of mass), Navier-Stokes equations for the three dimensions (conservation of momentum) and the energy equation (conservation of energy). The results are the fields of pressure, temperature and velocity. This solver implements the SIMPLE algorithm, which makes the pressure-velocity coupling by the Poisson equation, whose description can be found in Patankar and Spalding (1983).

2.5 Coupling with optimization

The entire project is divided in three parts: data reduction (statistical analysis), CFD (Computational Fluid Dynamics) and optimization. The packages used in each part are Software R, OpenFOAM and Dakota respectively. The statistical part is responsible for analyzing the data from the tests and provide the boundary conditions and numerical validation. The optimization coupled with numerical simulation is responsible for two processes: calibrate the temperature of the pipe external surface and perform all configurations using this temperature in order to find the one which provides the best thermal efficiency.

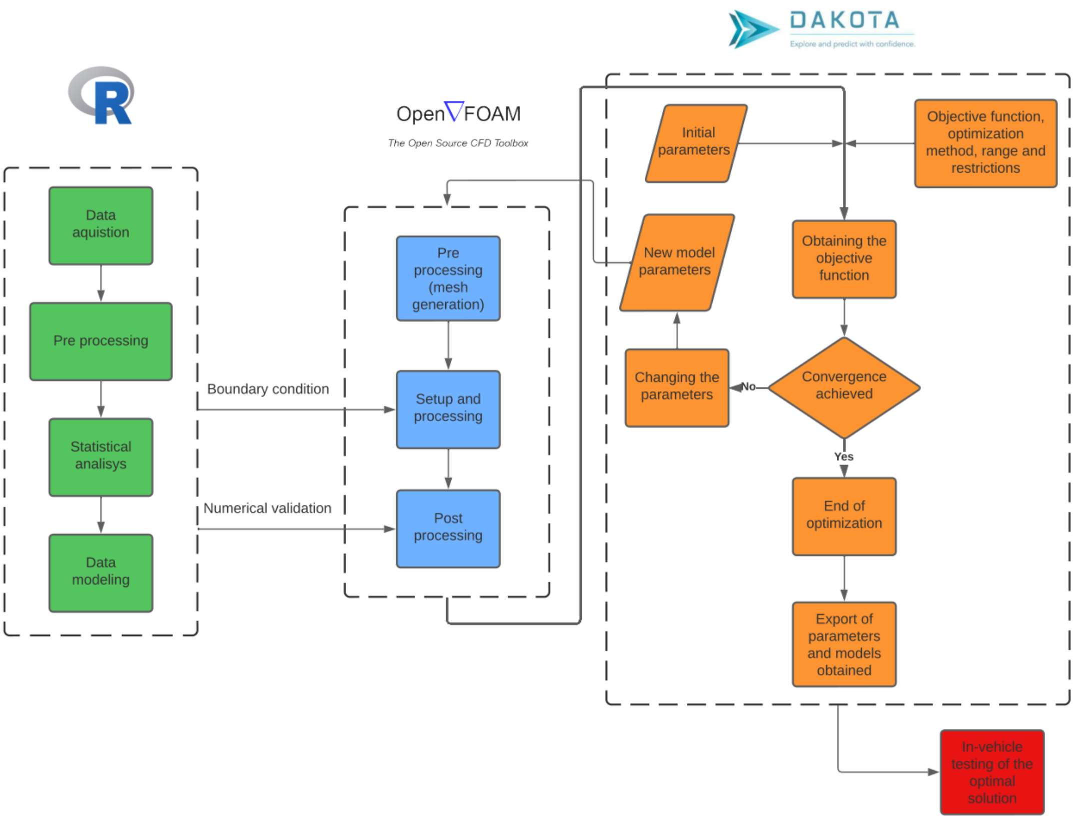

Figure 4. Workflow of the entire project

For the optimization of the temperature of the external surface, an algorithm based on gradient was used (Rao (2019)).

This method was chosen because it is a convex function, i.e., the algorithm tends to reach the global minimum easily, in addition to saving time.

3. RESULTS AND CONCLUSION

Using the data provided by statistical analysis (Tab. 4 and Fig. 5) for the reference pipe (without insulation), the temperature in the external surface was found through an optimization process.

Table 4. Data from statistical analysis

Ambient temperature [K]	306.71
Inlet temperature [K]	309.76
Outlet temperature [K]	309.81
Flow rate [m³/s]	$3.27 * 10^{-4}$

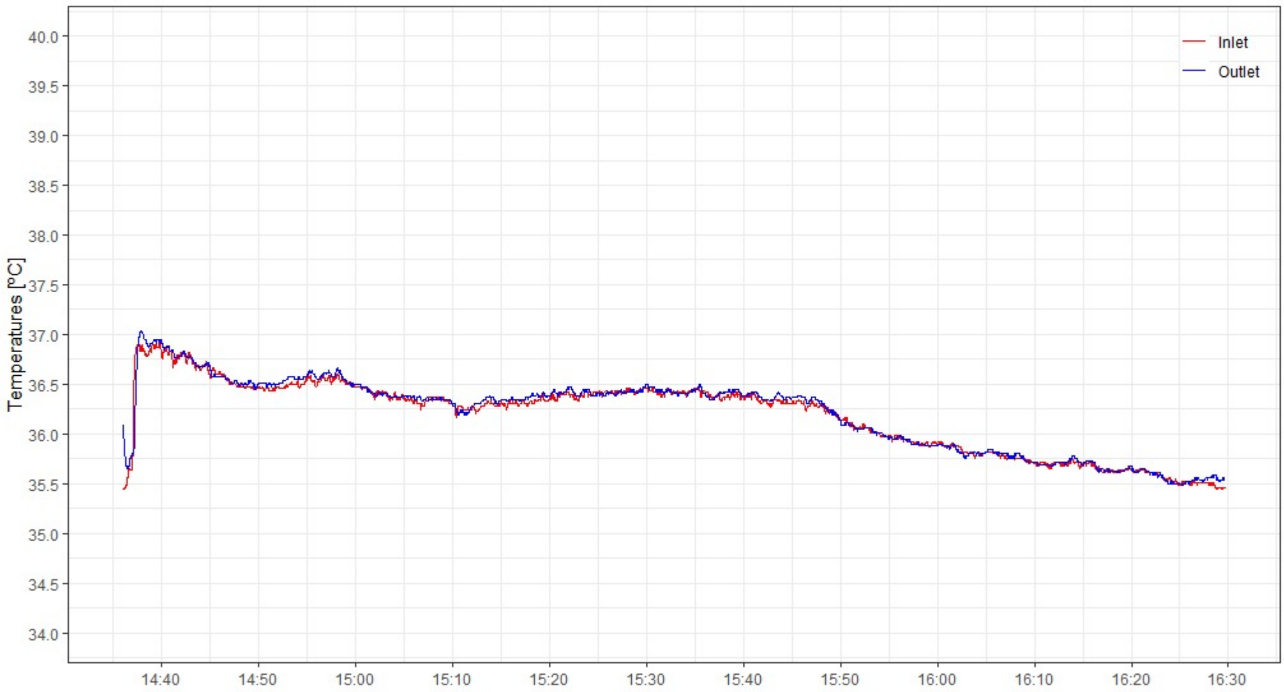

Figure 5. Monitoring of inlet and outlet temperatures

Using this temperature, it was possible to determine the performance of each insulation individually (Tab. 5).

Table 5. Performance of each insulation

Property	Thin insulation	Thick insulation
Temperature difference [K]	0.047	0.026
Reduction in temperature difference	12.8%	51.7%

These results show that thick insulation has a greater capacity to reduce the temperature difference between inlet and outlet. This effect is due to its greater thickness, given that the thermal conductivity are close. However, for smaller spaces, thin insulation may be a viable option.

Then, utilizing the same external temperature found by the fitting procedure realized, the proposed insulation were subjected to the optimization process using a genetic algorithm to obtain the best relation between cost and effectiveness. The specifications for the optimal insulation layer solution, along with the total cost and temperature differential between the inlet and outlet, can be found in Tab. 6. It was determined that to achieve a better balance between cost and effectiveness, the most efficient insulation layer arrangement consisted of four layers of thick insulation with no additional layers of thin insulation. This choice was made due to the higher cost and lower thermal resistance of thin insulation compared to thick insulation, which rendered the latter more economically viable. Furthermore, with this configuration, it was already possible to significantly reduce the temperature differential (resulting in a 85% reduction), making this arrangement a satisfactory choice without the need for additional layers that would increase the total cost without significant improvements in temperature differential reduction.

Table 6. Optimized configuration

Temperature difference [K]	0.008
Total cost per meter	16.00
Number of thin insulation	0
Number of thick insulation	4

The methodology highlighted the robustness and adaptability of the developed coupling for similar problems, involving analysis of geometric, physical properties and boundary condition parameters. It successfully established a generic approach that holds promise for future studies in this field.

4. ACKNOWLEDGEMENTS

The authors are grateful to the Mechanical Engineering Department of São Paulo State University campus of Ilha Solteira, which provided the entire infrastructure to perform this research.

This work has been possible in function of the Teaching, Research and Extension Foundation of Ilha Solteira grants, process n 006/2022 (scientific inititation).

5. REFERENCES

Celik, I.B., Ghia, U., Roache, P.J. and Freitas, C.J., 2008. "Procedure for estimation and reporting of uncertainty due to discretization in cfd applications". *Journal of fluids Engineering-Transactions of the ASME*, Vol. 130, No. 7.

Chen, K., Song, M., Wei, W. and Wang, S., 2018. "Structure optimization of parallel air-cooled battery thermal management system with u-type flow for cooling efficiency improvement". *Energy*, Vol. 145, pp. 603–613.

Chen, K., Wu, W., Yuan, F., Chen, L. and Wang, S., 2019. "Cooling efficiency improvement of air-cooled battery thermal management system through designing the flow pattern". *Energy*, Vol. 167, pp. 781–790.

Jones, W.P. and Launder, B.E., 1972. "The prediction of laminarization with a two-equation model of turbulence". *International journal of heat and mass transfer*, Vol. 15, No. 2, pp. 301–314.

Karimi, G. and Li, X., 2013. "Thermal management of lithium-ion batteries for electric vehicles". *International Journal of Energy Research*, Vol. 37, No. 1, pp. 13–24.

Maliska, C.R., 2004. *Transferência de Calor e Mecânica dos Fluidos Computacional*, Vol. 2. Grupo Gen-LTC.

Menter, F.R., 1994. "Two-equation eddy-viscosity turbulence models for engineering applications". *AIAA journal*, Vol. 32, No. 8, pp. 1598–1605.

Mondal, B., Lopez, C.F., Verma, A. and Mukherjee, P.P., 2018. "Vortex generators for active thermal management in lithium-ion battery systems". *International Journal of Heat and Mass Transfer*, Vol. 124, pp. 800–815.

Patankar, S.V. and Spalding, D.B., 1983. "A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows". In *Numerical prediction of flow, heat transfer, turbulence and combustion*, Elsevier, pp. 54–73.

Rao, S.S., 2019. *Engineering optimization: theory and practice*. John Wiley & Sons.

Richardson, L.F., 1911. "Ix. the approximate arithmetical solution by finite differences of physical problems involving differential equations, with an application to the stresses in a masonry dam". *Philosophical Transactions of the Royal Society of London. Series A, Containing Papers of a Mathematical or Physical Character*, Vol. 210, No. 459-470, pp. 307–357.

Richardson, L.F. and Gaunt, J.A., 1927. "Viii. the deferred approach to the limit". *Philosophical Transactions of the Royal Society of London. Series A, containing papers of a mathematical or physical character*, Vol. 226, No. 636-646, pp. 299–361.

SE, B., 2022. "Glysantin graphs". Disponível em: `https://www.glysantin.de/sites/default/files/2019-06/GLYSANTIN%C2%AE%20Graphs.pdf`. Acess: 14 jul 2022.

Spark, W., 2022. "Climate comparison in ilha solteira and sevilla". Disponível em: `https://weatherspark.com/compare/y/34152~29768/Comparison-of-the-Average-Weather-in-Sevilla-and-Ilha-Solteira#Figures-Temperature`. Acess: 14 jul 2022.

Tete, P.R., Gupta, M.M. and Joshi, S.S., 2021. "Developments in battery thermal management systems for electric vehicles: A technical review". *Journal of Energy Storage*, Vol. 35, p. 102255.

Versteeg, H.K. and Malalasekera, W., 2007. *An Introduction to Computational Fluid Dynamics - The Finite Volume Method*, Vol. 2. Pearson Education Limited.

Wilcox, D.C., 1988. "Reassessment of the scale-determining equation for advanced turbulence models". *AIAA journal*, Vol. 26, No. 11, pp. 1299–1310.

Zhao, G., Wang, X., Negnevitsky, M. and Zhang, H., 2021. "A review of air-cooling battery thermal management systems for electric and hybrid electric vehicles". *Journal of Power Sources*, Vol. 501, p. 230001.

6. RESPONSIBILITY NOTICE

The authors are solely responsible for the printed material included in this paper.