RESEARCH ARTICLE

Parametric Study of Electronic Cooling by Means of a Combination of Crossflow and an Impinging Jet

YUNESKY MASIP MACÍA, ANGEL A. RODRÍGUEZ SOTO, SULEIVYS M. NUÑEZ GONZALEZ, AND JACQUELINE PEDRERA YANES

1Escuela de Ingeniería Mecánica, Pontificia Universidad Católica de Valparaíso, Quilpué 2430120, Chile
2Escuela de Ingeniería Química, Pontificia Universidad Católica de Valparaíso, Valparaíso 2340000, Chile
3Departamento de Ingeniería Química y Ambiental, Universidad Técnica Federico Santa María, Valparaíso 2390123, Chile
4Department of Thermal Sciences and Fluids, Federal University of São João del-Rei, São João del-Rei-Minas Gerais 36307-352, Brazil

Corresponding author: Yunesky Masip Macía (yunesky.masip@pucv.cl)

This work was supported in part by the Ministry of Science and Innovation, Government of Spain, through the CICYT D+I Program, under Grant DPI2008-05349.

ABSTRACT

This paper reports a parametric study’s experimental results based on the experiments’ design. No works in the literature have investigated parametrically experimental effects on electronic component cooling by combining an impinging jet and a channel flow configuration on the cooling of electronic components. This study analyzed five parameters experimentally to enhance the cooling process using statistical techniques. Additionally, the optimal configuration determined for the conventional cooling method using only the channel flow was compared. Three parameters are associated with the geometric configuration (height of the electronic component $h/H = 1/6, 1/3, 1/2$, jet diameter $D/H = 0.3, 0.4, 0.5$ and jet-component eccentricity $S/H = 0, 1/8, 1/4$), and the other two are related to the fluid flow (the Reynolds number based on the channel height and mean velocity of the stream $Re_H = 3410, 4205, 5000$ and the ratio between the jet and the channel mean velocities $U_j/U_m = 2.5, 3.75, 5$). The results show that the main parameters are statistically significant relative to heat transfer, with $Re_H$, $U_j/U_m$ and $h/H$ displaying the most significant amplitude of variation in response and increasing the Nusselt number by approximately 60%. The surface response models have shown a satisfactory fit with the experimental data, allowing, in a preliminary way, the minimum mechanical energy loss requirement to be identified to maximize up to 160% of the heat transfer. Specifically, the heat transfer enhancement is more significant for components of considerable height than other components. Heat transfer enhancement occurs at low mechanical energy loss when the velocity ratio decreases at the minimum channel Reynolds number at the maximum jet diameter and jet-component eccentricity.

INDEX TERMS

Optimization, electronic cooling, impinging jet, cross flow, design of experiments, response surface model.

I. INTRODUCTION

Several works have debated the effects and rapid evolution of the electronics industry [1] and the requirements to increase the clock speeds of electronic components to achieve a higher performance [2]. Research has focused on improving the heat transfer of these components [3], [4], [5]. Mainly, the application of cooling techniques for electronic equipment using air flows, dielectric liquid-cooled, nanofluids, thermoelectric effects, and liquid immersion systems, among others, allows and facilitates an improvement in thermal performance [6], [7].

It should be noted that this work focuses on the cooling of electronic components using air as a working fluid.
As mentioned in the literature, there are recent works where the use of other fluids has shown thermal properties superior to air. However, it does not prevent air from becoming one of the most widely used fluids in the electronics industry. It was known that some of its thermal capacities might be limited, but its ease of work, low system complexity, and the possibility of optimizing the cooling process. It was demonstrated in the present investigation.

Despite this, current trends and how the research work in this area has been analyzed. First, considering the ease of use, availability and simplicity of the design, the air-cooling technique has been one of the most commonly used in electronic cooling systems. Natural, mixed, or forced air convection is used to cool electronics in various industrial applications [8], [9], [10]. In the same way, the design parameters of electronic cooling systems, such as the geometry, fan work, and manufacture of the printed circuit board (PCB), are essential factors for selecting and designing a suitable cooling system. Specifically, using the conventional method that involves only a cross flow (CF) or an impinging jet in combination with channel cross flow (IJCF) [11], [12].

The conventional cooling CF method has been shown to have lower performance on electronic systems [13], [14]. Therefore, investigations conducted to obtain acceptable values in heat transfer have been necessary to increase the performance of fans that move the more excellent air flow used. However, higher air flows are required to increase the mechanical losses of the cooling systems, which is not the desired effect of designers in the electronics industry [15], [16]. In this sense, numerical and experimental studies [17], [18], [19] have focused on the air flow features around a component or an array of components to determine the optimal velocity of the cooling system. Furthermore, these authors developed uniformly numerical models and correlations for component heating and submitted them to several flow conditions [20], [21], [22].

Other studies, such as the one presented by [23], investigated heat transfer for innovative ribs of four different types installed on the side walls of microchannel. This work was based on the computational fluid dynamics technique. The authors analyzed the performance of these configurations showing that their value was significantly increased. Furthermore, the rectangular ribs have a minimum Nusselt number compared to other ribs. It is relevant for the analysis of complex geometric configurations.

Similarly, Ahmad et al. [24] numerically studied the thermal behavior of different microchannel heat sinks and compared their results with traditional and non-traditional designs. An important finding was that a new microchannel cool more effectively than rectangular microchannels or other geometries. This new design showed the highest value of thermal enhancement at Reynolds number of 1000 and the highest thermal transport efficiency of almost 97%. Also, the work presented by [25] novelty evaluated the walls of a microchannel heat sink with various geometric configurations to improve its thermal and hydraulic characteristics.

The study was performed numerically using ANSYS Fluent and compared the performance of different microchannel heat sink configurations. The results of this study showed that the best performance was achieved with the configuration in which the rib spacing becomes equal to the bead length by 0.4 mm. The authors performed a model validation of the Nusselt number with experimental results of the literature.

One of the conventional ways to improve thermal performance has been using liquid-cooling [26], [27], [28]. These authors reported that the improvement in heat transfer was considerable, demonstrated by the high convection heat transfer coefficients. Recently, more studies of nanofluids have been undertaken among the fluids. They have improved heat removal rates in various thermal systems and impinging jet configurations [29], [30], [31]. However, one of the main disadvantages of nanofluids is that their thermal properties degrade over time because of sedimentation. In addition, this liquid-cooling with higher density and viscosity experiences a more significant pressure drop in the thermal systems where they have been used [32]. Some of these disadvantages have perpetuated the use of air-cooling.

Recently, studies using air flows have been focused on IJCF flow configurations over a heated component or array of components [33], [34]. The majority of these studies were developed numerically. Precisely, it will determine the heat transfer coefficients and compare a traditional channel flow configuration. The IJCF configuration provides high heat transfer rates in the stagnation region where the impinging jet impacts directly. However, the main disadvantage was the high energy consumption in these flow configurations [35]. These aspects improved the flow and heat transfer characteristics that have been extensively investigated over the last few years [36], [37], [38].

The authors presented relevant results on impinging jet use [39], [40]. In the first study, the authors analyzed the cooling of a flat plate under conditions of constant heat flow 1000 W/m², inside a rectangular channel, which was studied experimentally and numerically using a single air jet flow. Results show that the local and average Nusselt number was determined as the jet-to-plate distance function. Also, the Nusselt number depends on the Reynolds number based on the hydraulic diameter of the slotted nozzle based on a range for the speed of the outlet jet. This work showed that heat transfer is sensitive to Reynolds number and increases the average Nusselt number by 49.5%. However, the local Nusselt number was less sensitive to the change in the distance of the jet from the plate. Likewise, the numerical results fit well with the experimental data for local and average Nusselt numbers. Finally, the study proposed some correlations to predict the local Nusselt number.

The second study was based on the numerical investigation of the flow field of triangular and square fluted surfaces under a rectangular array of circular impinging air jets. Similarly [36], the effect of the Reynolds number of the jet and the jet-to-plate distance was examined, adding another parameter for the rib geometry on the fluid flow characteristics.
Among the most important results, the turbulence kinetic energy distribution and jet flow separation for square rib surface before impact were shown. Furthermore, jet separation occurred at a higher location with a high inlet velocity on the square ribbed surfaces. These fluid flow processes are essential in the effects of heat transfer. Many of these flow field structures positively or negatively affect the heat transfer enhancement from the heated surface. For this reason, it is essential to know these fluid behaviors [41].

Other authors [42], [43] have presented experimental work on improving heat transfer in cylindrical electronic components. These works determined the Nusselt numbers on the surface of the component and the pressure coefficients around it using a fully developed turbulent flow regime in the impinging jet. The results compared the effect of heat transfer for different heights of the component concerning a prismatic component, which was 80% higher than the latter. Similarly, in the recent work by [44], the authors experimentally and numerically investigated the effects of an array of impinging jets. This study analyzed the effect of different parameters on heat transfer. The results indicated that the heat transfer on the target surface increased with the length and diameter of the air duct. In addition, an average Nusselt number correlation was provided from the experimental results predicting the heat transfer within the ranges of the parameters studied.

An experimental study presented by [41] determined the Reynolds number of the channel and the relationship between the Reynolds numbers of the jet and channel on the Nusselt number and the pressure loss. The case studied was a single cubic component of dimension 15 mm for flow configurations of CF and IJCF. The authors showed that high Reynolds number ratios substantially improve component cooling performance. In the same way, the experimental results are compared with the literature data for a conventional CF method. However, this improvement in heat transfer of the IJCF configuration implies an increase in mechanical energy loss. Parametric experimental studies of this problem, that is, offering detailed data on the increase in the performance of the cooling process simultaneously with the analysis of pressure drops around the component, are scarce in the literature. Almost no information is available for this IJCF flow configuration regarding the optimal conditions that improve cooling and minimize mechanical system losses in electronic cooling.

Due to the current problem regarding the little information on optimal combinations of the design parameters, the scarce predictive models and the need for experimental results to study the IJCF configuration, the present work sets its research. The main objective of the present study was to investigate the influence of the five design parameters; namely, the Reynolds number of the channel, the ratio between the jet and the channel mean velocities, the height of the component, the diameter of the jet and the jet-component eccentricity in the upstream direction. Moreover, this study evaluated the enhancement of the heat transfer of an immersed electronic component in an IJCF reducing the mechanical power loss. Specifically, through an experimental parametric study using the Design of Experiments (DOE) statistical technique, the response surface models (RSM) for the Nusselt number and dimensionless pressure loss coefficient (response variables) are determined. The optimal combinations of the studied parameters were obtained through these models to optimize the response variable. Hence, the mathematical optimization and comparison for the conventional CF cooling method establish the optimal operating conditions for an electronic cooling system that uses an IJCF configuration using ambient air as the working fluid, minimizing the mechanical power losses.

The originality and novelty of the present study reside in optimizing the IJCF flow configuration for the cooling of electronic components. A parametric analysis determined the effects and main interactions of the parameters studied. In this way, a maximum heat transfer was obtained with a lower energy cost in the impulsion of the working fluid. In addition, the use of statistical techniques allows minimization of the cost and experimentation times, providing valid results. Likewise, surface temperature and pressure drop measurements were used as a power source for thermal calculations and head loss variables. Finally, the study was validated by comparing it to the classic cooling method and the numerical results found in the literature.

This study is an essential source for validating numerical models in this type of IJCF configuration. Additionally, the electronic system designers offer mathematical models that allow for high heat transfer with lower fan power requirements. Section 2 presented the methodology, describing the experimental and statistical procedure. Section 3 presents the main Results and Discussion about the five parameters that enhance heat transfer and mechanical power loss. In addition, the response surface model and optimization process are presented, as well as the comparison between the optimal combinations for IJCF for the conventional CF method. Finally, the study’s main conclusions are highlighted in the Conclusions section.

Electronic refrigeration systems employing an IJCF have been studied for various applications, as demonstrated in the Introduction. For this reason, the configuration itself is not an innovation. Instead, the innovative aspect lies in determining the optimal combination of geometric design parameters and the air flow used for cooling an electronic component. In addition, statistical techniques that have been little used for this type of experimental analysis were used in this work, which allowed for minimizing load losses, which produces lower economic costs in air delivery systems, maximizing heat transfer. Similarly, the proposed mathematical models allowed the designers of these electronic systems to make concrete decisions on which is the best option to cool the components. Additionally, the numerical models found in the literature were validated. Finally, another relevant aspect was the use of various advanced experimental techniques, as can be seen in the subsections of Experimental Setup and
Experimental Procedure, which offered accurate results for the scientific community.

II. METHODOLOGY

A. EXPERIMENTAL SETUP

The test rig used for the experimental measurements was designed and installed specifically to analyze and study the cooling of electronic components, as presented by the authors [41], [45]. However, the versatility of the experimental setup allowed the use of different experimental techniques for thermal and fluid field measurements. The experimental techniques are infrared thermography (IR), thermocouples and pressure drop measurements. In addition, different thermal and hydrodynamic conditions are generated in this setup, giving versatility to study electronic components cooling.

The electronic components were manufactured with the methodology proposed in the study of [41] or different heights of the components \( h_i / H \). According to the main objective, each component has a square section (characteristic length \( -L_c \)) equal to 15 mm or a variable component height. In this sense, the channel walls have the possibility of being assembled and disassembled with great ease to analyze different arrangements of electronic components, simulating a simple electronic system with elements mounted on PCB plates. Likewise, this experimental setup allows an impinging jet to be placed right on the opposite wall to which the component is mounted through a rigid tube long enough to obtain a fully developed flow. This plate can be changed to obtain different jet-component eccentricities in the upstream direction. It should be noted that the height of the channel \( (H = 2L_c) \) and the width are kept constant. Fig.1 shows a simplified representation of the experimental configuration and main dimensions of the test rig and the details of the thermal and pressure measurements.

B. DESIGN OF EXPERIMENT

As aforementioned, the conventional cooling of electronic components, that is, a component located in a channel and cooled only under the action of a cross flow has a limited efficiency due to flow conditions. For this reason, the present study adds an incident jet to the flow configuration, thus increasing the ability to cool the component. However, there is almost no information about the optimal conditions that improve cooling and minimize mechanical losses in the system.

In the study, several variables are analyzed to obtain the optimal combinations of the parameters for the heated component minimizing the mechanical power losses and maximizing the heat transfer. This work involves geometric and flows operating parameters, which is the main difficulty in the many experiments to be performed if all the possible combinations of the parameter values are tested.

Performing so many experiments would require time that is not ordinarily available. The question lies in selecting specific experiments to provide information as much as possible about the phenomenon being analyzed. This topic was addressed using the design of experiment techniques. The DOE statistic technique allows the organization and performance of the experiments has an advantage over the traditional ones because it requires fewer experiments to obtain the same precision on the estimation [46]. Consequently, the response variable predictive models and optimal values can be determined in the above cases. An example of the DOE was the study presented by [47]. The authors used an experimental design based on RSM to identify the effects of pin-fin heatsink design parameters on thermal performance. In experiments, they explored various design parameters that affect thermal performance. A relevant aspect of the experimental plan was applying a standard RSM design denominated Central Composite Design (CCD). The results distinguish the significant influencing factors to minimize two main variables. These aspects are relevant as a reference for the evaluation proposed in this work. Furthermore, using intentional changes in these variables makes it possible to identify the reasons for the changes in the investigated process [48]. In addition, others optimization techniques have been used successfully to analyze a microchannel heat sink through experimental studies [49], [50], [51], [52].

In this work, the DOE methodology developed by the research team of the Organization Department of TECNUN (University of Navarra) was followed and based on a 7-phase model [53]. The response variable was a variable obtained from the measurements due to the experiments in which the independent parameters that affect it are modified. These parameters are evaluated at different values during experimentation, and the expected results are validated once the experiment was finished. Therefore, literature studies [37], [54], [55] have been taken for the selection of parameters that produce substantial changes in heat transfer and the hydrodynamics of the field flow around the component.

Five dimensionless parameters were defined in the parametric study. Table 1 shows the values of the parameters. Two correspond to the fluid flow and the other three to the geometry used in the IJCF configuration. In terms of fluid flow, the parameters to be studied are the Reynolds number for channel flow \( (Re_H) \) and the ratio between the jet and the channel mean velocities \( (U_j / U_m) \). The geometric parameters are the height of the component \( (h / H) \), the diameter of the jet \( (D / H) \) and the jet-component eccentricity in the upstream direction \( (S / H) \). In the present parametric study, the height of the channel and the component width (characteristic length \( -L_c \) ) are held fixed. A schematic representation of each parameter for the parametric study is presented in Fig.2 for the \( XY \)-plane at \( Z / h = 0 \).

Therefore, using DOE, the parametric study was efficient regarding the number of experiments. Moreover, it was possible to state the conditions in which the experiments were performed. In this case, a fractional factorial design (FF) [56], [57] was used considering two values (levels) of the five parameters (factors), including five central points (each parameter at its intermediate value), so that 21 experiments...
were performed. Including the central points in the DOE allowed us to estimate the curvature in the response, that is, to obtain the response surface model. From this, the optimal values of the response variables are determined. In addition, the RSM allows us to predict the responses value in the range tested. The statistical software standard MINITAB (version 17.0) was used to build and analyze the selected design. Table 1 shows the experimental design created based on the five main parameters and their levels, considered typical of electronic refrigeration applications [58].

Once the parameters were defined, it was essential to propose the response variables determined from the experimental measurements, which correspond to thermal and hydrodynamic variables. First, the average Nusselt number on each of the component faces and the global number ($\bar{N_u}_m$), which quantifies the cooling of the electronic components and was determined from the area-weighted average heat transfer coefficient ($\bar{h}$). The latter was obtained from the surface temperature distributions and heat flux conduction. In addition, the Nusselt number [59] was defined using the characteristic length ($L_c$) in Equation (1)

$$\bar{N_u}_m = \frac{\bar{h}L_c}{k_{air}} \tag{1}$$

where $k_{air}$ is the thermal conductivity of the air.

The other variable response to consider was the dimensionless pressure loss coefficient ($K$), which was determined by first calculating the mechanical energy loss ($\dot{W}_L$) produced in each experiment, a control volume in the test section was
considered and applying the integral form of the mechanical energy equation [2], according to Equation (2)

\[ K = 2 \frac{W_L}{(\bar{m}_j + \bar{m}_e) U_m^2} = \ldots \left( \frac{\bar{m}_j + \bar{m}_e}{\rho \text{air} U_m^2} \right) \left[ \bar{m}_e P_{T,e} + \bar{m}_j P_{T,j} - (\bar{m}_e + \bar{m}_j) P_{T,out} \right] \]  \tag{2}

where \( P_T \) is the total pressure sum of the static and dynamic pressures, \( U_m \) corresponds to the mean velocity of the channel flow, \( \rho \) is the air density and \( \bar{m}_c \) and \( \bar{m}_j \) refer to the channel’s mass flow rate and the jet, respectively.

### TABLE 1. Details of the parametric study values of the parameters considered and the experimental sequence.

| Parameter (Factor) | Levels  |
|-------------------|---------|
| \( Re_H \)        | Low (-) | Middle (0) | High (+) |
| \( U_m \)         | 3410    | 4205    | 5000    |
| \( h/H \)         | 1/6     | 1/3     | 1/2     |
| \( D/H \)         | 0.3     | 0.4     | 0.5     |
| \( S/H \)         | 0       | 1/8     | 1/4     |

### C. EXPERIMENTAL PROCEDURE

The measurements were made in each experiment. Following the procedure described in [41], three components that simulate the thermal behavior of an actual electronic component were manufactured and the methodology to perform thermal measurements using IR thermography and T-type thermocouples. The thermal measurements under a steady state with convection from the air flows are performed to analyze the pressure losses. It was defined that the heat losses are secondary heat losses not due to convection from the air flows.

Jointly with the thermal measurements, pressure taps were manufactured and the methodology to perform thermal measurements of this work, described in detail by [41], are capable of recording the information and translating it into a potential signal read by the data acquisition cards. However, during this process, some errors are made which create an uncertainty in the measurements [63].

In this regard, [64] state that the total uncertainty (\( \delta R \)) in the measurement of a simple variable (\( R \)) is affected by the systematic (\( B_R \)) and random (\( A_R \)) errors and is estimated as shown the Equation (3) proposed by [63],

\[ \delta R = \left( (B_R)^2 + (A_R)^2 \right)^{1/2} \]  \tag{3}

In calculating a variable, measuring more than one parameter is necessary to obtain the final answer. In these cases, there is a propagation of uncertainty. Therefore, for a magnitude \( R \) which depends on \( n \) magnitudes (\( X_i \)), the total uncertainty of the response variable (\( \delta R \)) is obtained by Equation (4), [64],

\[ \delta R = \left( \sum_{i=1}^{n} \left( \frac{\partial R}{\partial X_i} \delta X_i \right)^2 \right)^{1/2} \]  \tag{4}

Starting from Equation (2), the simple variables on which the total uncertainty in \( Nu_{mn} \) depends are known, which are...
TABLE 2. Response variables were obtained from thermal and pressure drop measurements.

| Exp. | $N_{\text{m}}$ | $K$ | $W_{L}$ |
|------|----------------|-----|---------|
| 1    | 49             | 6.1 | 0.123   |
| 2    | 39             | 1.1 | 0.014   |
| 3    | 51             | 4.6 | 0.051   |
| 4    | 86             | 5.8 | 0.125   |
| 5    | 84             | 11.9| 0.247   |
| 6    | 69             | 11.0| 0.251   |
| 7    | 27             | 1.3 | 0.032   |
| 8    | 49             | 4.6 | 0.050   |
| 9    | 51             | 4.5 | 0.048   |
| 10   | 42             | 6.1 | 0.021   |
| 11   | 44             | 1.6 | 0.018   |
| 12   | 39             | 12.1| 0.079   |
| 13   | 40             | 2.2 | 0.010   |
| 14   | 58             | 1.9 | 0.052   |
| 15   | 18             | 1.8 | 0.038   |
| 16   | 32             | 2.2 | 0.010   |
| 17   | 51             | 4.6 | 0.047   |
| 18   | 49             | 6.2 | 0.018   |
| 19   | 72             | 11.8| 0.074   |
| 20   | 45             | 2.3 | 0.061   |
| 21   | 50             | 4.6 | 0.050   |

the conductivity of the epoxy ($k_{\text{epx}}$), the temperatures in the copper core of the component ($T_{\text{Cu}}$), the environment ($T_{\text{surr}}$), the input to the test section ($T_{\text{ref}}$) and component surfaces ($T_{\text{sup}}$), as well as the emissivity ($\varepsilon$) and the thickness of the epoxy layer ($t_{\text{epx}}$). The uncertainty was between $\pm$0.3% and $\pm$5.5% for all the analyzed cases. The minimum contributions to the uncertainty in the average Nusselt number correspond to the ambient and reference temperatures.

In the same way as the average Nusselt number, the uncertainty in the head loss coefficient was calculated. Among the simple variables on which the uncertainty of $K$ depends are the volumetric air flow rates measured in the laminar flow elements and the flow meter located in the pipe that produces the incident jet, as well as the geometric dimensions of the channel ($H$) and the jet tube ($D/H$). Its uncertainty value was between $\pm$1.0% and $\pm$2.6% for all experiments.

III. RESULTS AND DISCUSSION

A. PARAMETRIC STUDY

The results obtained through the DOE, defined in this study, show the main effects and interactions between the parameters for the average Nusselt number and the dimensionless pressure loss coefficient. In addition, these results determined the response surface model used to optimize each response variable.

Fig.4 shows the normal probability plots of the standardized residuals and the residues vs. prediction corresponding to the models for $N_{\text{m}}$ and $K$. From these graphs, each model’s accuracy and the model fit goodness can be assessed. The graph of probability plots of the standardized residuals shows
that most of the residuals for $Nu_m$ and $K$ are distributed along a straight line, which means that the effects are normally distributed and that not one of the 21 experiments has an absolute value greater than 2, which is the accepted value for identifying outliers [47], [56]. In the case of the graph of the residues vs. prediction, the points are distributed on both axes without a visible characteristic shape, indicating that in the experiments performed, no unusual values appear. In summary, it can be concluded that the determined RSM fit the experimental data satisfactorily.

The main effects of the five parameters considered on the average Nusselt number in each face and the entire component, in general, are represented in Fig.5 a). The Reynolds number for channel flow ($Re_H$), the ratio of velocities ($U_j/U_m$) and the height of the component ($h/H$) are the parameters that have the most significant influence on $Nu_m$. Increasing the first two parameters increases the average Nusselt number by approximately 50% and 61%, respectively. The component height ($h/H$) must be at its lowest level (1/6) and present the highest values for Num. Decreasing the component height ($h/H$) causes an increase in the average Nusselt number of approximately 33%.

Analyzing the component faces individually, the $Re_H$ improves the $Nu_m$ number to a similar degree, significantly higher in the top face ($T$). However, an increase of 68% in $Nu_m$ was achieved when the $U_j/U_m$ increased. In the frontal face ($F$) and side faces ($S$), heat transfer increases with an increase in $U_j/U_m$, similar to that shown by the $Re_H$ parameter, with an average increase of up to 55% in $Nu_m$. In the back face ($B$), the increment of $Nu_m$ is smaller than in the remaining faces of the component. In the case of the component height ($h/H$), increasing its value only increases the $Nu_m$ on the top face. The distance between the jet and the upper face decreases with increasing $h/H$, which produces a more significant effect on the jet in eliminating heat on this face.

In contrast, an increase in $h/H$ causes $Nu_m$ to decrease on the back face joint, with more modest decreases occurring on the front and side faces. It contributes to a decrease in the total average value of $Nu_m$ in all the components. In general, an increase in the levels of all the parameters causes the top face to present the highest values of the average Nusselt number observed, which is clearly due to the presence of the impinging jet. It is due to the complex effects of the flow field structures around the component (lateral vortices, recirculating bubble, wake vortex, lower horseshoe vortex) on heat transfer. The study by [41] shows that the rotational behavior of these vortices causes a longer residence time for the fluid in the vortex, allowing the local temperature of the fluid to increase. In this way, the local heat transfer by convection is reduced in that part of the faces of the component when it has greater height. In addition, for a higher $h/H$, areas of low fluid velocities appear near the rear face. Hence the little movement of the air current produces an increase in the local temperature of that region and fact, decreases local heat transfer by convection.

One of the options that could be considered in future work is using turbulence promoters similar to [65]. This work studied the effects of vortex promoters on cooling various flash-mounted electronic components, which have constant heat fluxes. First, the authors determined the enhancement heat transfer of the electronic components for effects of the length, location, number, and angular position of the vortex promoter for a Reynolds number and a relation between the distance of the jet to the plate and the hydraulic diameter of the nozzle used. A critical finding was that the heat transfer improved when the promoter vortex was closer to the jet entrance. Furthermore, it was observed that heat transfer is sensitive to the vortex promoter’s location, length and angular position. In addition, vortex promoters can be used to manipulate the flow field and improve thermal performance in electronic cooling. In the present work, observing how some flow structures organized around the component can discharge negatively in the enhancement of heat transfer was possible.

On the other hand, the effects of the diameter ($D/H$) and the jet-component eccentricity ($S/H$) on the average Nusselt number are less significant than those of the other three parameters mentioned above. As seen in Fig.5 a), the increase in both parameters produces a slight increase in the average Nusselt number. It is because heat transfer enhancement on the component was determined for a global behavior. The Nusselt number on the top face does not change appreciably for either parameter compared to the effect of the first three parameters. Instead, the Nusselt numbers on the front, side, and back faces similarly increase with the diameter. With the jet-component eccentricity in the direction upstream of the center of the component, the Nusselt number increases on the front and side faces and decreases on the back face.

The main effects were investigated for the pressure losses, similar to the analysis of the Nusselt number. Hence, Fig.5 b) presents the five parameters main effects on the dimensionless pressure loss coefficient. The results clearly show that only the ratio of velocities ($U_j/U_m$) and the jet diameter ($D/H$) substantially affect the coefficient. An increase in both parameters implies more effective jet air flow rates, which produces a higher coefficient value. The increase in $K$ is approximately 400% when the velocities ratio increases from the lower to the upper level. For the jet diameter parameter ($D/H$), the increase in the $K$ coefficient is almost 85%. The channel Reynolds number ($Re_H$), for its part, does not have a substantial influence on the dimensionless coefficient of pressure loss, but it has a notable influence on the pressure loss, which is proportional to the square of the flow velocity at the entrance to the channel, see (Equation 2), and therefore to the square of the Reynolds number. The component height ($h/H$) and the jet-component eccentricity ($S/H$) have a limited effect on the dimensionless coefficient of pressure losses as the $K$ coefficient does not show a clear trend when these parameters are varied.

The effects of the parameter interactions on $Nu_m$ and $K$ were investigated by analysis of variance (ANOVA)

103756

VOLUME 10, 2022
according to [47], [56]. In the case of interactions, their effect is determined by adding the parameter values with the same sign and subtracting when they differ in sign, dividing over half of the experiments performed. An interaction shows how the result of one parameter depends on the level at which the other was set. The p-value of each interaction was found using MINITAB software, and it was considered a significant interaction for those cases where a result less than 0.05 was obtained. Four significant interactions on the average Nusselt number are those between the channel Reynolds number \( (Re_H) \) and the ratio of velocities \( (U_j/U_m) \), the channel Reynolds number \( (Re_H) \) and the height of the component \( (h/H) \), the ratio of velocities \( (U_j/U_m) \) and the height of the component \( (h/H) \) and between the ratio of velocities \( (U_j/U_m) \) and the jet-component eccentricity \( (S/H) \) of the jet. In the case of the dimensionless pressure loss coefficient \( (K) \), only significant interaction was calculated between the ratio of the velocities \( (U_j/U_m) \) and the jet diameter \( (D/H) \).

From the previous results, the graph in Fig.6 was obtained for the significant interactions of \( Nu_m \) and \( K \). As the objective of the work is to enhance the heat transfer, with an increase in \( (Re_H) \) and \( (U_j/U_m) \), higher values of \( Nu_m \) were obtained. Similarly, the second significant interaction indicates that \( Nu_m \) also increases for values smaller than \( (h/H) \) and more significant than \( (Re_H) \). The following interaction shows that the smaller the height of the component \( (h/H) \)
and the greater the ratio \((U_j/U_m)\) is, the more it is possible to obtain higher average Nusselt numbers. Regarding the jet-component eccentricity \((S/H)\), an increase in this parameter combined with a higher ratio of velocities \((U_j/U_m)\) produces a significant enhancement in the heat transfer rate. Although the objective of the work is to reduce the mechanical power losses in the IJCF configuration, this first analysis offers a reference framework for achieving the proposed objective.

Following the same criteria, it was determined that the only significant interaction in the dimensionless pressure loss coefficient is between the ratio of velocities \((U_j/U_m)\) and the jet diameter \((D/H)\). The graph in Fig. 6 shows the significant interaction, which indicates that both parameters must be at their lowest levels to minimize the coefficient \(K\).

**B. RESPONSE SURFACE MODEL AND OPTIMIZATION**

The analysis of the most significant parameters and interactions makes it possible to obtain the response surface or prediction model based on the experiments performed. This model is nothing more than a polynomial equation derived from the data that expresses the relationship between the response and the essential parameters (and interactions). These models estimate and optimize the response in the regions of interest. To be considered a valid model, the coefficient of determination \(R^2_{adj}\) must be more than 75%, and the standard error of the regression \((S)\) must be as close to zero as possible [47], [56]. The models goodness of fit was established from these statistics to explain the results.

Using MINITAB software through ANOVA, all the values of the coefficients that define the models were obtained for the \(Nutm\) and \(K\). Finally, the prediction models or response surface models were obtained from the most significant parameters and interactions presented in the previous section.

The response surface models are expressed by Equation (5) and Equation (6) for \(Nutm\) and \(K\), respectively:

\[
Nutm = 49 + 9.59Re_H + 12.1 \left( \frac{U_j}{U_m} \right) - 6.39 \left( \frac{h}{H} \right) + 5.28 \left( \frac{D}{H} \right) + 5.01 \left( \frac{S}{H} \right) + 1.25 \left[ Re_H \cdot \left( \frac{U_j}{U_m} \right) \right]^2 - 2.01 \left[ Re_H \cdot \left( \frac{h}{H} \right) \right] - 5.30 \left[ \left( \frac{U_j}{U_m} \right) \cdot \left( \frac{h}{H} \right) \right] + 1.91 \left[ \left( \frac{U_j}{U_m} \right) \cdot \left( \frac{S}{H} \right) \right] \tag{5}
\]

\[
K = 5.30 + 3.50 \left( \frac{U_j}{U_m} \right) + 1.60 \left( \frac{D}{H} \right) + 1.22 \left[ \left( \frac{U_j}{U_m} \right) \cdot \left( \frac{D}{H} \right) \right] \tag{6}
\]

For (3), \(R^2_{adj} = 96.6\%\) and \(S = 0.05\), and for (4), \(R^2_{adj} = 99.5\%\) and \(S = 0.02\). The models obtained above can predict the average Nusselt number and the pressure loss coefficient within the limits of the studied factors. Moreover, it was validated with the normal probability plots of the residuals for \(Nutm\) and \(K\) displayed in Fig. 4 a) and b), respectively. Further, it supports the adequacy of the least-square fit. These models have two components: a linear part, which is associated with the main effects, and a nonlinear term, which corresponds to the interactions between the studied parameters.

The RSM indicates the influence of each parameter and significant interaction, whether it should increase or decrease to improve the response. In addition, the coefficient that multiplies each parameter and interaction in the models shows which parameter has a more significant or lower effect on the response variable \(Nutm\) or \(K\), respectively.

In Fig. 7, the experimental results obtained for \(Nutm\) and \(K\) are compared with the values predicted from the RSM and a second-order model using the literature data from a numerical study [2]. Similar parameters were used in this study but with ranges of values more exhaustive than those of the present study, which allows a statistically valid comparison. This comparison determined a difference of less than 8% in the fit for the experimental values for the average Nusselt number between the response surface models. This result was an acceptable value if it is known that the comparison is between a model of second-order and another of lower order. Likewise, the difference between the models and the experimental results in \(K\) is less than 10%. This process demonstrates the goodness of fit between the experimental data obtained and the RSM predictions. Therefore, the response surface model is valid for optimizing \(Nutm\) and \(K\) variables.

Table 3 shows the optimal combinations of the parameters for obtaining the highest \(Nutm\) and \(K\); these variables extreme points are analyzed individually.
The optimal combination of parameters for the average Nusselt number does not correspond to the optimal combination for the dimensionless coefficient $K$. Therefore, a consensus was determined between the values of the parameters studied to obtain a combination that provided the optimal cooling of the component for a given power of the pressure drops. These optimal combinations of the parameters were determined by applying the Standard Interval Global Engine (SIGE) method, described by [66], which was implemented in the Premium Solver for Microsoft Excel for the response surface models. This method was used to solve the global optimization problem (maximizing the average Nusselt number) required in this study, starting from a restriction for a given power loss. The average Nusselt number obtained with these optimal combinations was compared in the next section with the values obtained for component cooling using the conventional CF method. This comparison allowed the evaluation of the component cooling enhancement using the IJCF configuration.

C. COMPARISON BETWEEN THE CONVENTIONAL METHOD CF AND THE IJCF CONFIGURATION

The cooling of the electronic component using the conventional CF method was experimentally studied independently, according to the procedures described in [41]. The experiments required for the CF method were not included in the DOE parametric study, so they required an additional experimental campaign to determine $Nu_{im}$ and $K$. In this case, the values of the channels Reynolds number ($Re_H$) are determined by the flow limitations of the test rig (see [45]). Using the study previous results [39] for $Re_H$ of 3410, 5752 and 8880 at a component height of $h/H = 1/2$, it was only necessary to experiment with $Re_H$ values of 4205 5000. In addition, the component height of $h/H = 1/6$ was...
included in the experiments for all values of $Re_H$. In this way, conventional CF refrigeration was compared with the case of IJCF studied by DOE. Importantly, no measurements were made in the CF configuration for the component height of $h/H = 1/3$ because this height corresponds to the central point experienced in the parametric study, which only has statistical validity for the DOE techniques.

Fig. 8 shows the average Nusselt number and the power loss obtained for the conventional CF method in terms of the Reynolds number of the channel. In the comparison between both cooling methods, the dimensionless power loss was calculated for the reference power ($\dot{W}_{ref} = \dot{m}_c U_m^2/2$) considering the case of the highest Reynolds number of the channel used in the DOE ($Re_H = 5000$). As Fig. 8 shows, both variables $Nu_m$ and $\dot{W}_L/\dot{W}_{ref}$, increase with $Re_H$, and a higher value of the average Nusselt number with a lower power loss was obtained with a smaller component height ($h/H = 1/6$).

![Figure 8](image-url) **FIGURE 8.** Experimental results of the conventional CF method: (a) average Nusselt number and (b) dimensionless power loss.

Because of the previous results for the conventional cooling CF method, to make a detailed comparison with the IJCF method, the average Nusselt number was obtained from the optimal combinations of the five parameters studied. Each optimal combination for $Nu_m$ was found from the procedure described in the previous section applying the SIGE method. The global optimization problem was resolved to maximize $Nu_m$ with a given power loss restriction. Once the optimal combinations for $Nu_m$ were found at each component height for the IJCF configuration, they were compared with the average value obtained using the conventional CF method. Fig. 9 compares the cooling methods, in which the $Nu_m$ values for the case of IJCF determined from the optimal combinations were represented by the blue line. For component heights ($h/H$) of 1/6 and 1/2, the enhancement in the cooling of the component using the IJCF configuration was more considerable than in the case of the conventional CF method (black line). Mainly, for the highest component $h/H = 1/2$, the increase in the average Nusselt number was presented and compared to that of the lowest component $h/H = 1/6$. At the latter height, the enhancements obtained in the cooling of the component, due to the optimal combinations, were not as considerable compared with the actual results of the experiments performed in the parametric study (red line). Comparing each component height shows that when $h/H = 1/2$, there is a higher $Nu_m$ increase between the CF case and the values found in the DOE to the lower height component $h/H = 1/6$. The average Nusselt number is always higher for the minor component without comparing it with other cases.

In Fig. 9, the points in IJCF configuration indicate the changes in the parameters through which the maximum $Nu_m$ values are obtained (see the table to the right of the graph in Fig. 9). In the case of the lower component ($h/H = 1/6$) between points 1 and 2, the optimal combinations of the parameters were obtained by increasing the value of the velocity ratio ($U_j/U_m$) without varying the values of the remaining parameters. The optimum is obtained from points 2 to 3 by increasing the parameters $Re_H$ and $D/H$ until they are above point 3, and the optimal combinations are only obtained by increasing $Re_H$ to 4760. In the case of the most considerable component height ($h/H = 1/2$), all optimal combinations are achieved by individually increasing a parameter (see the table of values to the right of the graph in Fig. 9). Between points 5 and 6, for dimensionless power of the lowest head losses, optimal cooling is achieved with increasing Reynolds number of the channel. As the dimensionless power loss was increased to acceptable values, the jet diameter ($D/H$) and the velocity ratio ($U_j/U_m$) were increased to maintain optimum cooling (between points 6 and 8). All optimal combinations were obtained at both component heights by keeping the jet-component eccentricity ($S/H$) at the maximum value of 1/4.

Based on these results, when comparing the average Nusselt number calculated by the optimal combinations, between points 1 and 4 for $h/H = 1/6$, the enhancement in the component cooling is approximately 45%, with the cost of considerably increasing power loss up to 500%. The same result occurs for the highest component $h/H = 1/2$. However, comparing the optimal combinations of the IJCF with the conventional CF method, for example, in the case of the near point between the blue and black curves (points 1 and 1’ in Fig. 9, respectively) for a constant power loss, the increment of $Nu_m$ is approximately 100%. For the greater component height of $h/H = 1/2$, an effect similar to the previous case occurs. Comparing points 5 and 5’ of Fig. 9 shows that, for a constant power loss, the enhancement in the average Nusselt number was approximately 160% between the IJCF and CF. However, lower values of the average Nusselt number than those of the component with the lowest height ($h/H = 1/6$).

Therefore, to produce a significant enhancement in the rate of heat transfer with a minimum power loss in a higher electronic component submitted to the IJFC configuration, the Reynolds number of the channel $Re_H$ and the velocity ratio $U_j/U_m$ must be at their lowest value, while the diameter $D/H$ and jet-component eccentricity $S/H$ must be at their highest value. On the other hand, for lower height components immersed in an IJCF configuration, improvements in cooling are obtained when the Reynolds number of the channel and the diameter of the jet are at their lowest value, while the velocity ratio and the jet-component eccentricity should be at their highest value.
Finally, it was verified that the response surface models determined using (3) and (4) provide a valid result for the average Nusselt number and the dimensionless pressure loss coefficient. This verification was demonstrated by selecting an optimal combination of the parameters and experimenting with these values. First, the selection of the optimal combination was defined from the results found previously for a value located between points 5 and 6 of Fig. 9 in the case of a higher component height of \( \frac{h}{H} = \frac{1}{6} \), where a minimum power loss was achieved. Then, the combination of the parameters was introduced in the models, and the \( \text{Nu}_m \) and \( K \) were predicted. Table 4 shows the results obtained from the experimentation and the mathematical models using the optimal combinations. The differences between the average Nusselt number obtained experimentally and the predicted value were approximately 4%. However, the dimensionless pressure loss coefficient determined by the response surface model was 2.3% lower than that obtained experimentally.

### IV. CONCLUSION

An experimental parametric study was performed to investigate the effect of five parameters on enhancing heat transfer and pressure loss for electronic cooling using IJCF. The average Nusselt number and dimensionless pressure loss coefficient over the electronic component were obtained for different \( Re_H \) \( (Re_H = 3410, 4205, 5000) \), \( \frac{U_j}{U_m} \) \( (\frac{U_j}{U_m} = 2.5, 3.75, 5) \), \( \frac{h}{H} \) \( (\frac{h}{H} = 1/6, 1/3, 1/2) \), \( \frac{D}{H} \) \( (\frac{D}{H} = 0.3, 0.4, 0.5) \) and \( \frac{S}{H} \) \( (\frac{S}{H} = 0, 1/8, 1/4) \), using a heat flux of 2500 W/m². The experimental parametric study was developed using the DOE and RSM statistical techniques. The following conclusions can be drawn from the experimental results:

- On \( \text{Nu}_m \), the parameters \( Re_H \), \( \frac{U_j}{U_m} \) produce an increase between 50% and 61%, and the decrease of 33% in \( \frac{h}{H} \) produce an increase of \( \text{Nu}_m \).
The optimization process allows a comparison between the response surface models, resulting in differences between the experimental and predicted results lower than 4% for both $N_u m$ and $K$.

**ACKNOWLEDGMENT**

The authors would like to thank the support of Escuela de Ingeniería Mecánica, Pontificia Universidad Católica de Valparaíso, and the Department of Thermal Sciences and Fluids, Federal University of São João del-Rei, Minas Gerais, Brazil.

**REFERENCES**

[1] E. P. DeBenedictis, “It’s time to redefine Moore’s law again,” *Computer*, vol. 50, no. 2, pp. 72–75, Feb. 2017, doi: 10.1109/MC.2017.34.

[2] G. S. Larraona, A. Rivas, R. Antón, J. C. Ramos, I. Pastor, and B. Moshfegh, “Computational parametric study of an impinging jet in a cross-flow configuration for electronics cooling applications,” *Appl. Therm. Eng.*, vol. 52, no. 2, pp. 428–438, Apr. 2013, doi: 10.1016/j.applthermaleng.2012.12.027.

[3] B. Indulakshmi and G. Madhu, “Heat transfer modeling and simulations for electronic cooling systems embedded with phase changing materials,” *Heat Transfer-Asian Res.*, vol. 47, no. 1, pp. 185–202, Jan. 2018, doi: 10.1002/htrj.21298.

[4] E. R. Meinders and K. Hanjalić, “Experimental study of the convective heat transfer from in-line and staggered configurations of two wall-mounted cubes,” *Int. J. Heat Mass Transf.*, vol. 45, no. 3, pp. 465–482, 2002, doi: 10.1016/S0017-9310(01)00180-6.

[5] B. C. Fejó, G. Lorenzini, L. A. Isoldi, L. A. O. Rocha, J. N. V. Goulart, and E. D. D. Santos, “Constructal design of forced convective flows in channels with two alternated rectangular heated bodies,” *Int. J. Heat Mass Transf.*, vol. 125, pp. 710–721, Oct. 2018, doi: 10.1016/j.ijheatmasstransfer.2018.04.086.

[6] C. M. Hussain, *Handbook of Nanomaterials for Industrial Applications*. Part V: Electronics and Electrical Industry, 1st ed. Cambridge, ON, Canada: Elsevier, 2018, doi: 10.1016/C2016-0-04427-3.

[7] B. B. Kanbur, C. Wu, S. Fan, and F. Duan, “System-level experimental investigations of the direct immersion cooling data center units with thermodynamic and thermoeconomic assessments,” *Energy*, vol. 217, Feb. 2021, Art. no. 119373, doi: 10.1016/j.energy.2020.119373.

[8] S.-F. Chou and I.-P. Tsern, “Mixed convection heat transfer of horizontal channel flow over a heated block,” in *Transport Phenomena in Heat and Mass Transfer*, J. A. Reizes, Ed. Amsterdam, The Netherlands: Elsevier, 1992, pp. 492–503, doi: 10.1016/B978-0-444-89851-7.50048-0.

[9] R. C. Chu, R. E. Simons, M. J. Ellsworth, R. R. Schmidt, and V. Caizolino, “Review of cooling technologies for computer products,” *IEEE Trans. Device Mater. Rel.*, vol. 4, no. 4, pp. 568–585, Dec. 2004, doi: 10.1109/TDMR.2004.885855.

[10] H. M. Maghrabie, M. Attalla, H. E. Fawaz, and M. Khalil, “Impingement/effusion cooling of electronic components with cross-flow,” *Appl. Therm. Eng.*, vol. 151, pp. 199–213, Mar. 2019, doi: 10.1016/j.applthermaleng.2019.01.106.

[11] S. Bünia and M. Tehranipoor, “Printed circuit board (PCB): Design and test,” in *Hardware Security*. Amsterdam, The Netherlands: Elsevier, 2019, pp. 81–105, doi: 10.1016/B978-0-12-812477-2.00009-5.

[12] J. Zhu, R. Dou, Y. Hu, S. Zhang, and X. Wang, “Heat transfer of multi-slot nozzles air jet impingement with different Reynolds number,” *Appl. Therm. Eng.*, vol. 186, Mar. 2021, Art. no. 116470, doi: 10.1016/j.applthermaleng.2020.116470.

[13] H. M. Maghrabie, M. Attalla, H. E. Fawaz, and M. Khalil, “Effect of jet position on cooling an array of heated obstacles,” *J. Thermal Sci. Eng. Appl.*, vol. 10, no. 1, Feb. 2018, Art. no. 011005, doi: 10.1115/1.4036788.

[14] A. Meghdip, T. Benabdallah, and A. Dellil, “Impact of geometry of electronic components on cooling improvement,” *Int. J. Heat Technol.*, vol. 37, no. 1, pp. 167–178, Mar. 2019, doi: 10.18280/ijht.370121.

[15] Q. Yu, Z. Mei, M. Bai, D. Xie, Y. Ding, and Y. Li, “Cooling performance improvement of impingement hybrid synthetic jets in a confined space with the aid of a fluid diode,” *Appl. Therm. Eng.*, vol. 157, Jul. 2019, Art. no. 113749, doi: 10.1016/j.applthermaleng.2019.113749.

[16] D. H. Lee, Y. S. Chung, and P. M. Ligrani, “Jet impingement cooling of chips equipped with multiple cylindrical pedestal fins,” *J. Electron. Packag.*, vol. 129, no. 3, pp. 221–228, Sep. 2007, doi: 10.1115/1.2753884.

[17] T. Ming, C. Cai, W. Yang, W. Shen, and T. Gan, “Optimization of dimples in microchannel heat sink with impinging jets—Part A: Mathematical model and the influence of dimple radius,” *J. Thermal Sci.*, vol. 27, no. 3, pp. 195–202, Jun. 2018, doi: 10.1016/j.jts.2018.02.009.
NOTE: The natural text appears to be a collection of abstracts and references from various scientific papers. It seems to be focused on heat transfer and fluid dynamics, particularly involving impinging jets and their applications in cooling systems. The references cover a wide range of topics, from nanofluids and their properties to the design of heat sinks and the use of experimental and computational methods to study these phenomena.

For example, a paper by M. Kilic, T. Calisir, and S. Baskaya titled "Experimental and numerical study of heat transfer from a heated flat plate in a rectangular channel with an impinging air jet" was published in the Brazilian Journal of Mechanical Sciences, 2019. Another paper by J. Bogus and K. Ahmad, "Experimental investigation of the effect of a surfactant to increase cooling of high temperature steel surface," was published in the Journal of Heat Transfer, 2019.

The references also include studies on the optimization of heat sinks using various algorithms and techniques, such as the Jaya algorithm for optimizing a micro-channel heat sink. There are also discussions on the use of nanofluids, particularly alumina nanofluid-based air atomized spray impingement, and the influence of dimple height and arrangement on heat transfer.

Overall, the papers seem to be focused on improving cooling techniques in electronic and mechanical systems, with a particular emphasis on the use of impinging jets and the optimization of heat sinks.
YUNESKY MASIP MACÍA received the Ph.D. degree in industrial engineering from the University of Navarra, Spain, in 2014. He is currently a Full Professor and a Ph.D. Supervisor with the School of Mechanical Engineering, Pontificia Universidad Católica de Valparaíso, Quilpué, Chile. His research interests include the development of mathematical models for the material behavior, computer-aided mechanical design and manufacturing, biomechanical design, and composites development and modeling.

ANGEL A. RODRÍGUEZ SOTO received the B.S. and M.S. degrees in mechanical engineering from the Technological University of Havana, Havana, Cuba, in 2007 and 2013, respectively, and the Ph.D. degree in mechanical engineering from the Technological University of Havana and the Federal University of Rio de Janeiro, Brazil, in 2017. From 2007 to 2018, he was a Professor and a Researcher with the Technological University of Havana. He is currently a Professor with the School of Mechanical Engineering, Pontificia Universidad Católica de Valparaíso. His research interests include the development of mathematical models for the material behavior, computer-aided mechanical design and manufacturing, biomechanical design, and composites development and modeling.

SULEIVYS M. NUÑEZ GONZALEZ was born in 1988. She received the bachelor’s degree in chemical engineering from the Technological University of Havana, Havana, Cuba, in 2011, and the Ph.D. degree in biotechnology from the Pontifical Catholic University of Valparaíso and Technical University of Federico Santa María, Valparaíso, Chile, in 2020. She is currently an Associate Researcher with the Technical University of Federico Santa María. Her research and academic interests include statistics techniques and environmental sciences. In addition, she has collaborated with energy technologies.

JACQUELINE PEDRERA YANES was born in 1971. She received the Ph.D. degree in mechanical engineering from the Polytechnical School of São Paulo University, São Paulo, Brazil. She is currently a Full Professor with the Department of Thermal and Fluid Sciences, Federal University of São João Del-Rei. Her current research and academic interests include energy technologies, design of experiments, fluid mechanics, and transport phenomena.