

Evaluation of the thermal performance of an Organ-on-a-Chip device: a numerical study

Mariana Ferreira^{1*}, Glauco Nobrega^{1,2}, Lucas Neves¹, Carlos Faria¹, Ana Moita^{3,4}, Diana Pinho^{5,6}, Rui A. Lima^{1,7,8}

1: Mechanical Engineering and Resource Sustainability Center (MEtRICs), Mechanical Engineering Department, University of Minho, Campus de Azurém, 4800-058 Guimarães, Portugal.

2: Centro de Investigação de Montanha (CIMO), Campus de Santa Apolónia, Instituto Politécnico de Bragança, 5300-253 Bragança, Portugal.

3: IN+, Center for Innovation, Technology and Policy Research, Instituto Superior Técnico, Universidade de Lisboa, Av. Rovisco Pais, 1049-001 Lisboa, Portugal.

4: CINAMIL—Centro de Investigação Desenvolvimento e Inovação da Academia Militar, Academia Militar, Instituto Universitário Militar, Rua Gomes Freire, 1169-203 Lisboa, Portugal.

5: Center for MicroElectroMechanics Systems (CMEMS), University of Minho, Campus de Azurém, 4800-058, Guimarães, Portugal.

6: Associate Laboratory in Biotechnology and Bioengineering and Microelectromechanical Systems (LABBELS) – University of Minho, Braga/Guimarães, Portugal.

7: Transport Phenomena Research Center (CEFT), Faculdade de Engenharia da Universidade do Porto (FEUP), Rua Roberto Frias, 4200-465 Porto, Portugal.

8: Associate Laboratory in Chemical Engineering (ALiCE), Faculty of Engineering, University of Porto, 4200-465 Porto, Portugal.

* Correspondent author: id12157@uminho.pt

Keywords: Organ-on-a-chip, Microfluidics, Heat transfer, Laminar flow

ABSTRACT

Conventional 2D and 3D culture models and animal testing fail to accurately reproduce human physiological complexity, limiting the predictive capability of preclinical studies. Organ-on-a-Chip (OoC) platforms have emerged as a promising alternative by enabling precise control of the cellular microenvironment. In this work, a microfluidic system incorporating an integrated heating strategy for OoC applications was developed and analyzed, aiming to maintain physiological temperature conditions ($\sim 37^\circ\text{C}$). Computational fluid dynamics (CFD) simulations were performed using ANSYS Fluent to investigate fluid flow and heat transfer within the device. The microfluidic device was designed to contain an inlet, a main circular reservoir, and an outlet, connected by narrow channels. A heating boundary condition of 37°C was imposed on the external bottom wall of the device, while the fluid entered at room temperature (25°C) at a flow rate of $500\ \mu\text{L}/\text{h}$. A mesh sensitivity study was conducted using three hexahedral meshes comprising 9 599, 30 658, and 99 440 elements. The medium mesh (30 658 elements) was selected as it provided the best balance between computational cost and accuracy, with an average orthogonal quality of 0.975 and an average skewness of 0.112, both classified as excellent. Steady-state simulations demonstrated stable and symmetric laminar velocity profiles throughout the geometry, with no recirculation zones observed. The flow field exhibited a parabolic cross-sectional profile in the channels and a radial deceleration pattern within the reservoir, consistent with low Reynolds number microfluidic behavior. Regarding thermal performance, temperature monitoring at five points along the device confirmed that regions downstream of the inlet (P2–P5) reached values between 36.943°C and 36.945°C .

°C under steady-state conditions, closely approaching the physiological target. Only the inlet point (P1) recorded a lower temperature (28.753 °C) due to the continuous entry of cooler fluid. Transient simulations at $t = 5$ s and $t = 200$ s confirmed rapid thermal stabilization: by $t = 200$ s, temperatures at P2–P5 reached 36.971–36.993 °C, demonstrating progressive convergence toward a highly uniform thermal regime. These results validate the proposed heating strategy as an effective and computationally efficient approach to maintaining physiological temperature in OoC platforms, supporting the development of more reliable in vitro models for biomedical research and drug screening applications.

1. Introduction

Traditional static 2D and 3D culture systems fail to faithfully replicate the complexity of human physiological responses (Huh et al., 2011). As a result, current preclinical models remain limited in their ability to predict unexpected side effects, hindering the development of effective treatments for life-threatening diseases such as cancer (Esch et al., 2015). The Organ-on-a-Chip (OoC) devices emerged as an alternative to the traditional culture methods, since these systems can mimic the characteristics of the physiological microenvironment realistically, allowing the understanding, investigation, and treatment of several diseases as different types of cancer or to understand the adverse effects of drugs in multiple organs (Carvalho, Banobre, et al., 2022). OoC systems have also gained attention due to their potential to reduce animal testing by providing more physiologically relevant human-based models for drug screening and disease modelling.

Precise control and monitoring of key physiological parameters in OoCs, including temperature, O_2 and CO_2 concentrations, and pH, are achieved through the integration of microscale channels, tissue scaffolds, dynamic culture conditions, and embedded sensors (Carvalho, Rodrigues, et al., 2022). Among these parameters, temperature plays a particularly crucial role in maintaining consistent physical, chemical, and biological reactions, making stable thermal regulation essential for cell viability (Ahmed, 2013; Kwon & Jung, 2023). Maintaining temperatures close to 37 °C is essential for reproducing physiological conditions and ensuring proper mammalian cell function and viability. Even slight temperature fluctuations may alter cell metabolism, enzyme activity, membrane permeability, proliferation rates, and gene expression, directly affecting the reliability and reproducibility of experimental results. However, maintaining homogeneous and stable temperature profiles in microfluidic devices remains challenging due to their small dimensions, high surface-area-to-volume ratio, and continuous fluid flow, which can lead to heat losses and temperature gradients. Different heating approaches have been explored in microfluidic platforms, including external heaters, resistive heating elements, and integrated thermal control systems. Numerical simulation represents an important tool in the design of

microfluidic systems, enabling the prediction of heat transfer and fluid dynamics behaviour before fabrication. This approach reduces experimental costs, accelerates optimization, and enables evaluation of multiple design parameters under controlled conditions. Therefore, the development of reliable thermal management strategies is essential for improving the performance and reproducibility of OoC platforms.

The main objective of this work is to develop a microfluidic system for OoC applications that includes a heating system to maintain a stable temperature (around 37 °C). To better understand the behavior of the temperature and the physical phenomena involved, the tests were done using numerical simulation. Computational evaluation allows not only to have a broader perspective on the fluid flow phenomena but also the characterization and optimization of the system under study with reduced time and resources.

2. Methods and materials

2.1. Geometry, mesh development and analysis

The developed geometry consists of an inlet and outlet, followed by a channel that leads to the main reservoir, as presented in Figure 1. The geometry was established based on the work developed in the group, Mariana et al., 2026 (Ferreira et al., 2026), simplifying the model to only one reservoir.

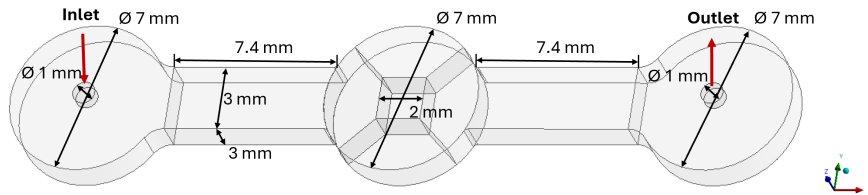


Figure. 1 3D details of the designed geometry and its main dimensions.

The mesh development is a crucial step in numerical simulation, since it represents the physical domain by dividing the entire geometry into smaller volumes. To improve the mesh quality even more, some slices were made along the geometry, as depicted in Figure 1. In the developed study, a mesh evaluation was done, analyzing the velocity at three points in three hexahedral meshes with different refinements. The 3 points were defined in interesting regions: after the inlet, in the middle of the reservoir, and before the outlet, as represented in Figure 2 (a). The three developed meshes include a coarse mesh, Mesh 1 with 9599 elements, a medium mesh, Mesh 2 with 30658 elements, and a fine mesh, Mesh 3 with 99440 elements.

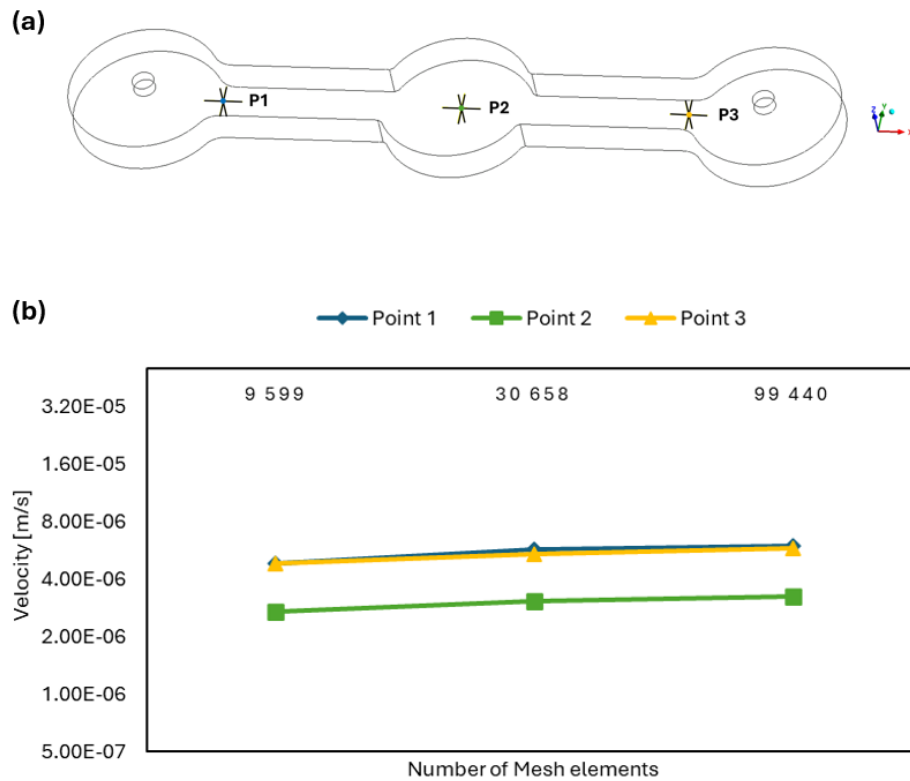


Figure. 2 Post-processing CFD: (a) points positioning; and (b) flow velocity at Points 1, 2, and 3 as a function of mesh elements.

From the analysis of Figure 2 (b), Mesh 1 was discarded compared to the two other meshes due to the significant velocity difference in all the analyzed points. Between Mesh 2 and 3, the results stabilized, and the differences are not substantial. Considering that Mesh 3 involves a higher computational cost, Mesh 2 was selected as it offered the best trade-off between performance and computational time.

The accuracy of computational results depends on mesh quality. To evaluate it, the orthogonal quality and skewness were used. This parameter is calculated for each cell based on its skewness and the vector connecting the cell centroid to each of its faces. Orthogonal quality ranges from 0 to 1, where 0 corresponds to the lowest quality and 1 to the highest. Skewness, in turn, indicates how close a cell or face is to the ideal shape. It also varies between 0 and 1, with 1 representing the poorest quality and 0 the best (*ANSYS Fluent User's Guide*, 2020). The developed geometry presents an average orthogonal quality of 0,97545 and an average skewness of 0,11225, indicating an excellent quality, as is possible to see in Figure 3.

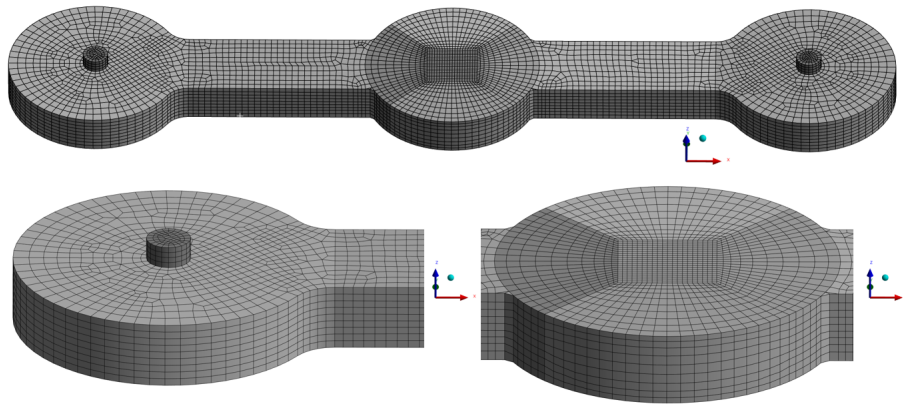


Figure. 3 Detailed mesh on microfluidic channels.

2.2. Boundary conditions

In the simulations, a laminar and incompressible flow regime was assumed. The working fluid was defined as a culture medium with a density of 1002 kg/m^3 and a dynamic viscosity of $8.48 \times 10^{-4} \text{ kg/(m}\cdot\text{s)}$ (Poon, 2020). Thermal properties were also considered for water, including a specific heat capacity of $4178 \text{ J/(kg}\cdot\text{K)}$ (The Engineering ToolBox, 2003) and a thermal conductivity of $0.6 \text{ W/(m}\cdot\text{K)}$ (The Engineering ToolBox, 2018). The solid domain, representing the device material (PDMS), was modeled with a density of 965 kg/m^3 (Ullas et al., 2018), a specific heat capacity of $1460 \text{ J/(kg}\cdot\text{K)}$ (Nobrega et al., 2024), and a thermal conductivity of $0.15 \text{ W/(m}\cdot\text{K)}$ (Souza et al., 2025; Waheed et al., 2017).

The CFD model was implemented assuming steady-state and transient conditions for the flow field, while accounting for heat transfer between the fluid and solid domains. At the inlet, a uniform velocity of $3.089 \times 10^{-5} \text{ m/s}$ was imposed, corresponding to a flow rate of $500 \text{ }\mu\text{L/h}$ (Cho et al., 2016). The inlet and outlet temperatures were set to 25° (room temperature). A no-slip boundary condition was applied at all walls. To simulate thermal effects, a heating wall condition was imposed on the external surface of the PDMS, with a fixed temperature of 37° (physiological temperature conditions) (Suhendra et al., 2020). The wall thickness was defined as 2 mm , allowing the evaluation of heat transfer across the solid domain and its impact on the fluid temperature distribution.

2.3. Governing equations

For the numerical solution, ANSYS Fluent software was used, where the governing equations are solved using the finite volume method.

The Navier-Stokes equations ensure mass and momentum conservation and are presented in Equations (7) and (8) (Carvalho et al., 2024):

$$\nabla \vec{v} = 0 \quad (7)$$

$$\nabla(\rho \vec{v} \vec{v}) = -\nabla p + \eta \nabla^2 \vec{v} + \rho \vec{g} \quad (8)$$

Where ρ is the density, $\vec{v}$ is the velocity vector, p is the pressure, η is the dynamic viscosity, and $\vec{g}$ is the gravity vector.

3. Results

3.1. Numerical fluid flow evaluation - steady state and transient simulations

Providing an overview of the velocity field under steady-state conditions (Figure 4), the flow distribution in both the XY and XZ planes shows a consistent and symmetric pattern along the geometry. As expected, the velocity reaches its maximum along the central axis of the connecting channels, forming elongated high-velocity regions between the reservoirs. In contrast, near the walls, the velocity decreases significantly, approaching near-zero values due to the no-slip boundary condition.

At the transitions between the narrow channel and the wider circular reservoir, a clear deceleration of the flow is observed, with velocity spreading radially into the reservoir. This expansion leads to more uniform but lower velocities inside this zone. The inlet and outlet regions exhibit localized velocity peaks associated with the entry/exit of the fluid. In the XZ plane, the profile confirms a developed velocity distribution across the channel height, with a parabolic-like profile typical of laminar flow. Overall, the results align with theoretical expectations, showing smooth acceleration in constricted regions and deceleration in expanded zones, with no significant recirculation under steady-state conditions.

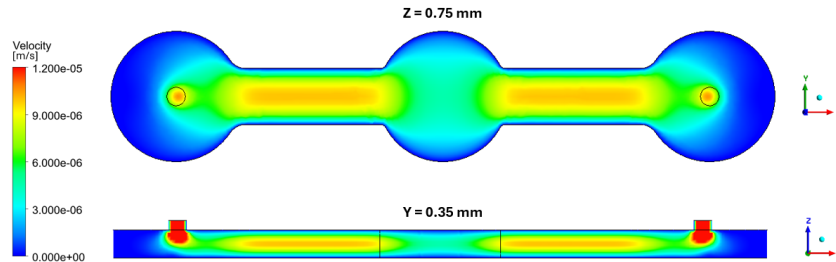


Figure. 4 Velocity fields in the geometry through CFD steady-state results at XY and XZ planes.

Analyzing the transient (Figure 5) behavior at $t = 5s$ and $t = 200s$, the velocity fields in both XY and XZ planes demonstrate that the flow rapidly stabilizes over time. Between both observed times, no visible differences were observed; the flow field is nearly identical to the steady-state

solution, indicating that a steady regime has been reached. The center channel regions maintain the highest velocities, while the reservoirs continue to show reduced speed due to flow expansion. The XZ plane further confirms that the velocity profile across the channel height stays stable, with the highest velocities in the central region of the channel. No significant recirculation zones are observed during the infection, and the flow remains stable and symmetric throughout the process.

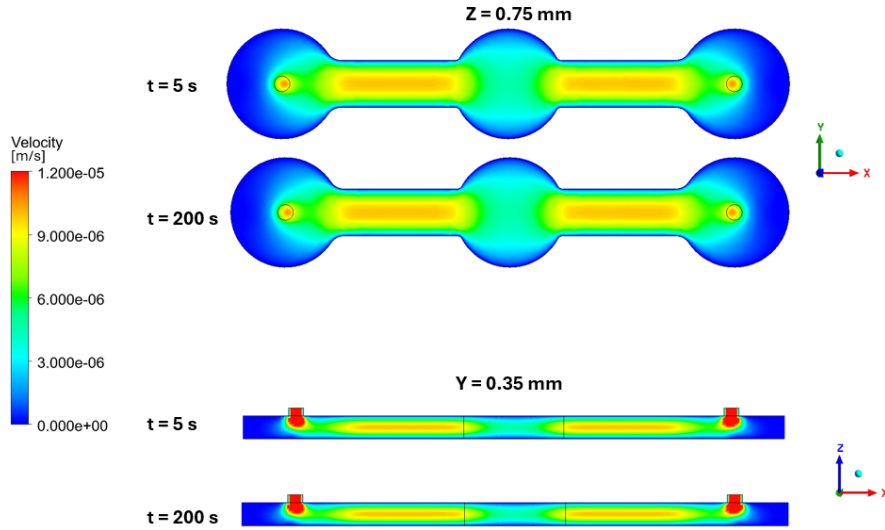


Figure. 5 Velocity fields in the geometry through CFD transient-state results (5s and 200s) at XY and XZ planes.

3.1. Heating distribution evaluation - steady state and transient simulations

For the first computational tests, the temperature was applied to the bottom wall. Providing an overview of the temperature distribution under steady-state conditions (Figure 6), the results in both XY and XZ planes show a predominantly uniform thermal field across the geometry. As expected, most of the domain reaches temperatures close to the imposed heating wall condition (37°), indicating efficient heat transfer through the PDMS and into the fluid.

However, a localized region of lower temperature is observed near the inlet, where the fluid enters at 25° . This creates a temperature gradient, with a clear thermal boundary layer forming in this region. As the fluid progresses along the channel, it rapidly equilibrates with the surrounding heated walls, resulting in a nearly homogeneous temperature distribution in the downstream reservoirs and channel sections.

In the XZ plane, the temperature profile across the channel height confirms that the fluid near the heated walls reaches higher temperatures, while slightly cooler values persist near the inlet region. Overall, the results indicate that under steady-state conditions, thermal equilibrium is largely achieved throughout the device, with only minor deviations near the inlet.

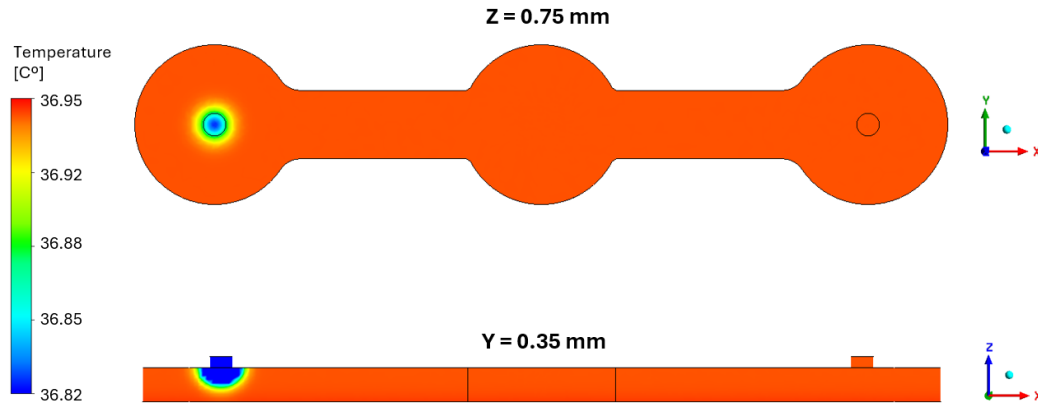


Figure. 6 Temperature field in the geometry through CFD steady-state results at XY and XZ plane.

Analyzing the transient temperature evolution at $t = 5$ s and $t = 200$ s, a clear progression of heat propagation along the geometry can be observed. At the early stage ($t = 5$ s), the temperature field shows a strong gradient near the inlet, where the cooler fluid (25°) contrasts sharply with the heated walls (37°). This results in a pronounced thermal front, with heat beginning to diffuse and convect into the fluid domain.

By $t = 200$ s, the temperature field evolves significantly, showing a clear propagation of heat from the inlet region toward the rest of the geometry. The upstream reservoir and adjacent channel sections remain cooler, while downstream regions approach the wall-imposed temperature. A distinct temperature gradient zone is visible between these regions, representing the progression of the thermal front.

The XZ plane further illustrates this behavior, showing the development of a temperature gradient across the channel height, with higher temperatures near the walls and lower temperatures in the core flow, particularly at earlier times. As time progresses, this gradient diminishes as the system moves toward thermal equilibrium.

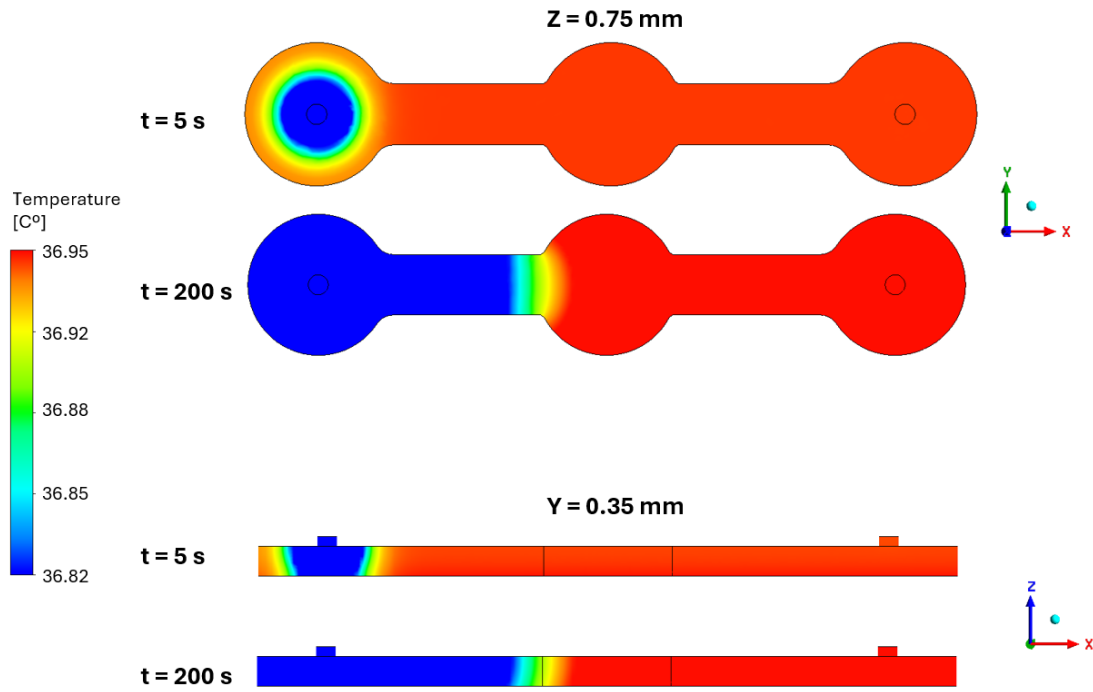


Figure. 7 Temperature field in the geometry through CFD transient-state results (5s and 200s) at XY and XZ planes.

Temperature values were obtained at the monitoring points to provide a quantitative evaluation of the thermal distribution within the geometry under both steady-state and transient conditions. The chosen points were selected in interest zones: inlet, outlet, and top, middle, and bottom parts of the reservoir, as presented in Figure 8. Overall, the results demonstrate that the heating system maintains a highly stable, homogeneous temperature distribution across most regions of the device.

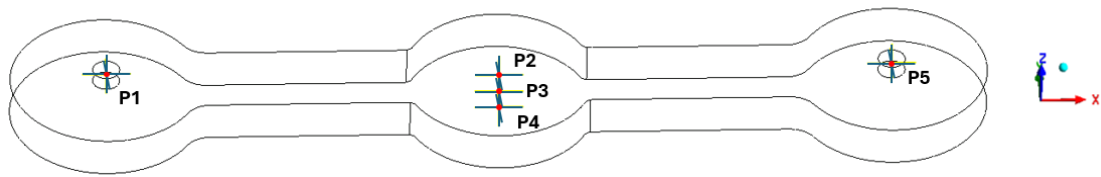


Figure. 8 Points for the temperature evaluation: in the inlet (P1), in the top of the reservoir (P2), in the middle of the reservoir (P3), in the bottom of the reservoir (P4), and in the outlet (P5).

Under steady-state conditions, the temperatures measured at P2, P3, P4, and P5 are highly consistent, ranging between 36.943 °C and 36.945 °C. These results indicate that the central and downstream regions of the geometry rapidly reach thermal equilibrium, maintaining temperatures very close to the target physiological range. In contrast, P1 presents a significantly lower temperature (28.753 °C), which is expected due to its proximity to the inlet region, where the cooler incoming fluid enters the device before being heated by the surrounding walls.

For the transient simulations, the results at $t=5$ s show that the downstream regions (P2–P5) already achieve temperatures close to the steady-state solution, with values around 36.944–36.949 °C. This indicates a rapid heat transfer process and efficient thermal conduction through the PDMS walls. However, P1 remains at a substantially lower temperature (27.078 °C), showing that the inlet region requires more time to thermally stabilize due to the continuous entrance of cooler fluid.

At $t=200$ s, a slight increase in temperature is observed at points P2–P5, reaching values between 36.971 °C and 36.993 °C. These results suggest that the system approaches a highly stable thermal regime over time, with minimal spatial temperature variations across the geometry. The small differences between the monitoring points confirm the uniformity of the heating process and demonstrate that the device is capable of providing near-constant thermal conditions in the regions of interest.

Table. 1 Temperature values in the defined points (P1, P2, P3, P4 and, P5)

		P1	P2	P3	P4	P5
Steady state		28.753°C	36.943°C	36.945°C	36.945°C	36.943°C
Transient state	5 s	27.078°C	36.944°C	36.945°C	36.949°C	36.944°C
	200 s	27.078°C	36.971°C	36.972°C	36.974°C	36.993°C

4. Conclusions and future work

The present work successfully demonstrated the development and numerical evaluation of a microfluidic OoC platform with integrated thermal control. CFD simulations performed under steady-state and transient conditions showed stable laminar flow behavior and efficient heat transfer throughout the device. The results obtained demonstrated that the proposed heating strategy was capable of maintaining a nearly homogeneous temperature distribution close to physiological conditions (around 37 °C) across most regions of the geometry, while transient simulations revealed a rapid thermal stabilization process.

The numerical approach proved to be an important tool for understanding the coupled fluid flow and heat transfer phenomena occurring inside the microfluidic device, enabling the optimization of the system with reduced experimental costs and development time. Overall, the developed platform demonstrates strong potential for OoC applications requiring controlled and reproducible thermal environments, contributing to the improvement of physiologically relevant in vitro models for biomedical research and drug screening applications.

Despite the promising results, some limitations remain, including the simplified geometry and the idealized thermal boundary conditions adopted in the simulations. Future work will therefore focus on improving the geometry by incorporating more complex and physiologically relevant features, as well as performing experimental validation of the numerical results. Additionally, further studies will be conducted to optimize the integration of the heating system, including the evaluation of alternative heating strategies, boundary conditions, and sensor integration for real-time thermal monitoring. In particular, future simulations will aim to more closely replicate the experimental setup, assessing whether temperature control should continue to be applied at the bottom wall or implemented through alternative heating configurations. The influence of different flow rates, materials, and device configurations on the thermal and flow behaviour will also be investigated to further improve the robustness and physiological relevance of the platform.

Acknowledgments

The authors are grateful to the Fundação para a Ciência e a Tecnologia (FCT), for partially financing the Project 16244 - 2023.18254.ICDT - Waste2NFs, 2022.06207.PTDC (<https://doi.org/10.54499/2022.06207.PTDC>), and 2023.17540.ICDT (<https://doi.org/10.54499/2023.17540.ICDT>) also partially funded by FERDER.. The authors also acknowledge the partial financial support by national funds through FCT/MECI/ MCTES (PIDDAC): UID/04077/2025; UID/00532/2025; UID/00690/20252023.18254.ICDT, UIDP/50009/2025, UIDB/50009/2025, and LA/P/0045/2020. A.S. Moita also acknowledges FCT for partially financing her contract through CEECINST/00043/2021/CP2797/CT0005, doi: 10.54499/CEECINST/00043/2021/CP2797/CT0005. Lucas Neves acknowledge the financial support by FCT, of the doctoral grant 2025.00900.BDANA. Glauco Nobrega was supported by the doctoral grant PRT/BD/153088/2021 (<https://doi.org/10.54499/PRT/BD/153088/2021>), financed by the Portuguese Foundation for Science and Technology (FCT), and with funds from MCTES/República Portuguesa, under the MIT Portugal Program. Mariana Ferreira acknowledges the financial support by FCT, of the doctoral grant 2025.00916.BDANA.

References

- Ahmed, K. (2013). *Treating cancer with heat : Hyperthermia as promising strategy to enhance apoptosis*. 63(April), 504–508.
- ANSYS *Fluent User's Guide*. (2020, July). <https://www.ansys.com/>
- Carvalho, V., Banobre, M., Minas, G., Teixeira, S., Lima, R. A., & Rodrigues, R. O. (2022). The integration of spheroids and organoids into organ-on-a-chip platforms for tumour research : A review. *Bioprinting*, 27, e00224. <https://doi.org/10.1016/j.bprint.2022.e00224>

- Carvalho, V., Gonçalves, I. M., Rodrigues, N., Sousa, P., Pinto, V., Minas, G., Kaji, H., Shin, S. R., Rodrigues, R. O., Teixeira, S. F. C. F., & Lima, R. A. (2024). Numerical evaluation and experimental validation of fluid flow behavior within an organ-on-a-chip model. *Computer Methods and Programs in Biomedicine*, 243. <https://doi.org/10.1016/j.cmpb.2023.107883>
- Carvalho, V., Rodrigues, N., Rodrigues, R. O., Teixeira, J. C., Miranda, J., Lima, R. A., & Teixeira, S. (2022). Influence of the Inlet Velocity on Oxygen Gradients in a Liver-on-a-Chip Model. *International Mechanical Engineering Congress & Exposition, ASME International Mechanical Engineering Congress and Exposition*. <https://doi.org/10.1115/IMECE2022-96001>
- Cho, S., Islas-Robles, A., Nicolini, A. M., Monks, T. J., & Yoon, J.-Y. (2016). In situ, dual-mode monitoring of organ-on-a-chip with smartphone-based fluorescence microscope. *Biosensors and Bioelectronics*, 86, 697–705. <https://doi.org/10.1016/j.bios.2016.07.015>
- Esch, E. W., Bahinski, A., & Huh, D. (2015). Organs-on-chips at the frontiers of drug discovery. *Nature Reviews Drug Discovery*, 14(4), 248–260. <https://doi.org/10.1038/nrd4539>
- Ferreira, M., Carvalho, V., Lima, R. A., Teixeira, S., & Pinho, D. (2026). Organ-on-a-chip platform with enhanced mixing capabilities: A numerical and experimental investigation. *Results in Engineering*, 29, 109440. <https://doi.org/10.1016/j.rineng.2026.109440>
- Huh, D., Hamilton, G. A., & Ingber, D. E. (2011). From 3D cell culture to organs-on-chips. *Trends in Cell Biology*, 21(12), 745–754. <https://doi.org/10.1016/j.tcb.2011.09.005>
- Kwon, S., & Jung, S. (2023). *Combination Therapy of Radiation and Hyperthermia , Focusing on the Synergistic Anti-Cancer Effects and Research Trends*.
- Nobrega, G., Souza, R., Cardoso, B., Afonso, I., Pereira, J., Cardoso, E., Moita, A., Ribeiro, J., & Lima, R. (2024). Experimental evaluation of green nanofluids in heat exchanger made of PDMS. *Thermal Science and Engineering Progress*, 55, 102978. <https://doi.org/10.1016/j.tsep.2024.102978>
- Poon, C. (2020). *Measuring the density and viscosity of culture media for optimized computational fluid dynamics analysis of in vitro devices*. <https://doi.org/10.1101/2020.08.25.266221>
- Souza, R., Nobrega, G., Afonso, I. S., Pereira, J., Cardoso, E., Marques, F., Vilarinho, C., Moita, A., & Lima, R. A. (2025). Thermal performance evaluation of pure PDMS and PDMS composites heat exchangers. *Journal of Thermal Analysis and Calorimetry*, 150(13), 10401–10412. <https://doi.org/10.1007/s10973-025-14394-3>
- Suhendra, N. F., Handayani, S., Budiawan, & Dani, I. C. (2020). *Effect of bisphenol A and Cu(I) exposure that induces oxidative stress through 8-OHdG formation in physiological conditions*. 020028. <https://doi.org/10.1063/5.0001079>
- The Engineering ToolBox. (2003). *Water Thermal Properties – Density, Heat Capacity, and Conductivity*.
- The Engineering ToolBox. (2018). *Thermal Conductivity of Water: Temperature and Pressure Data*.
- Ullas, A. V., Kumar, D., & Roy, P. K. (2018). Poly(dimethylsiloxane)-toughened syntactic foams. *Journal of Applied Polymer Science*, 135(8). <https://doi.org/10.1002/app.45882>
- Waheed, S., Cabot, J. M., Macdonald, N. P., Kalsoom, U., Farajikhah, S., Innis, P. C., Nesterenko, P. N., Lewis, T. W., Breadmore, M. C., & Paull, B. (2017). Enhanced physicochemical properties of polydimethylsiloxane based microfluidic devices and thin films by incorporating synthetic micro-diamond. *Scientific Reports*, 7(1), 15109. <https://doi.org/10.1038/s41598-017-15408-3>