

Numerical modelling of heat transfer during fluid flow in a mini annular channel using simcenter STAR-CCM+

Magdalena Piasecka^{1*}, Krzysztof Galiszewski¹, Beata Maciejewska² and Paweł Łabędzki²

¹ Kielce University of Technology, Faculty of Mechatronics and Mechanical Engineering, Al. 1000-lecia Państwa Polskiego 7, 25-314 Kielce, Poland

² Kielce University of Technology, Faculty of Management and Computer Modelling, Al. 1000-lecia Państwa Polskiego 7, 25-314 Kielce, Poland

Abstract. This study presents a numerical analysis of heat transfer during steady, single-phase flow of HFE-649 in a mini annular channel. Fully three-dimensional simulations were performed in Simcenter STAR-CCM+ to predict temperature, velocity, and pressure fields, as well as local variations of the heat transfer coefficient. The computational model represents a small vertical annular geometry corresponding to a laboratory test section that is currently being developed for future experimental validation. A three-dimensional model created in SolidWorks was imported into Simcenter STAR-CCM+, where a polyhedral mesh refined near solid–fluid interfaces was applied. The governing equations of mass, momentum, and energy conservation were solved. A laminar fluid flow model was adopted. The boundary conditions reflected typical parameters of compact heat exchange systems, including the prescribed mass flow rate, inlet temperature, outlet pressure, and constant wall heat flux. The simulations yielded detailed temperature, velocity, and pressure fields that characterise the thermal and flow behaviour within the channel. The obtained results provide a numerical reference for the design and calibration of the experimental setup and will serve as a numerical reference for validating and optimising mini annular channel configurations in compact heat exchangers.

1 Introduction

The study of heat transfer and fluid flow in mini annular channels is of growing importance in the design of compact and high-efficiency thermal systems. Such geometries are increasingly being applied in advanced heat exchangers, electronic component cooling modules, and energy systems where space and weight constraints are critical. Due to the small hydraulic diameter, flow and heat transfer processes in mini-scale channels differ significantly from those in conventional systems, requiring detailed numerical and experimental investigation. However, few studies have focused on the heat transfer in mini annular channels, which are of particular relevance to compact heat exchangers.

Reference [1] reports experimental results on heat transfer during turbulent single-phase water flow in smooth concentric annular channels. The study aimed to develop a correlation for the Nusselt number (Nu) using a 6 m long horizontal copper tube setup, with diameter ratios from 1.7 to 5.1 and Reynolds numbers (Re) between 4000 and 30000. Heat transfer coefficients were obtained using the Wilson method, and the proposed correlation predicted Nusselt numbers within $\pm 3\%$ of the experimental data.

Reference [2] presents experiments on convective heat transfer in a narrow annular channel, comparing

horizontal and vertical orientations. The concentric tube system had a hydraulic diameter of 4.12 mm and a length of 1500 mm. Both laminar and turbulent regimes were analysed. Reduced heat transfer efficiency was observed for $Re < 150$, while the laminar–turbulent transition occurred earlier ($Re \approx 800\div 1200$) than in conventional channels. In the fully turbulent regime, the narrow annulus enhanced heat transfer compared to standard geometries.

A combined experimental–numerical study examined single-phase water flow in a horizontal mini annular channel composed of 4.6 m long stainless steel tubes with a 3.0 mm gap. For Reynolds numbers between 1100 and 1729, simulations revealed a more uniform velocity distribution and lower peak velocities than in conventional channels. The Nusselt number increased with heat flux, Reynolds number, and inlet temperature, while the heat transfer coefficients were 1.3 to 2.1 times higher than in larger channels.

Computational Fluid Dynamics (CFD) provides an effective tool for analysing thermo–fluid behaviour under controlled boundary conditions and for predicting local temperature, velocity, and pressure distributions. Among the available CFD platforms, Simcenter STAR-CCM+ offers advanced capabilities for multiphysics modelling, mesh optimisation, and postprocessing of thermal and flow data.

* Corresponding author: tmpmj@tu.kielce.pl

Study [3] developed a liquid cooling system for lithium-ion batteries in electric vehicles using Simcenter STAR-CCM+. Six aluminium cooling plate designs with 3–13 water channels were modelled. Parametric analysis covered channel width, mass flow rate, heat flux, and inlet temperature. The best performance was achieved for configurations with five 18 mm channels and seven 16 mm channels.

References [4] and [5] focused on CFD modelling in STAR-CCM+. In [4], flow of HFE-649 was simulated through a 5×5 rod bundle with a spacer grid, showing good agreement with the experimental data and confirming the reliability of the model to predict the average flow properties. In [6], heat and mass transfer with thin film evaporation in a rectangular channel were analysed; the numerical results were consistent with experimental evaporation rates within 10%, validating the accuracy of the model.

In previous research, the authors conducted experimental investigations in the Laboratory of Boiling Heat Transfer at the Faculty of Mechatronics and Mechanical Engineering, Kielce University of Technology (KUT), using test modules with rectangular minichannels [6] and a test module with a mini annular channel [7]. Numerical analyses were performed using various computational methods, including approaches based on Trefftz functions and the ADINA software for comparative analyses [8], as well as the Homotopy Perturbation Method combined with Trefftz functions and the Simcenter STAR-CCM+ software [9]. However, the use of STAR-CCM+ was previously focused only on modules with rectangular minichannels.

This work focusses on the numerical simulation of single-phase forced convection in a mini annular channel using Simcenter STAR-CCM+. The main objectives are to determine the temperature, velocity, and pressure fields, assess local heat transfer characteristics, and prepare a validated numerical model that will support ongoing experimental research.

2 Numerical model of a mini annular channel

2.1 The simulation procedure

The modelling approach involved defining the physical domains, generating an appropriate computational mesh, specifying material properties, and setting the boundary conditions based on typical operating parameters of compact heat exchange systems. The governing equations of mass, momentum, and energy conservation were solved using the finite-volume method [10]. The details of mesh refinement, selected laminar flow model, and convergence criteria are presented in the following subsections.

2.2 Geometry and computational domain

The numerical domain represents a vertical mini annular channel formed between a cylindrical inner wall and an

outer glass wall. The dimensions were selected according to the experimental design to ensure realistic flow and heat transfer conditions. The model also includes inlet and outlet sections to minimise the influence of boundary effects on the main flow region.

A three-dimensional digital model of the test module used for numerical simulations in Simcenter STAR-CCM+ was created in SOLIDWORKS. The longitudinal cross-section of the test module is presented in Fig. 1.

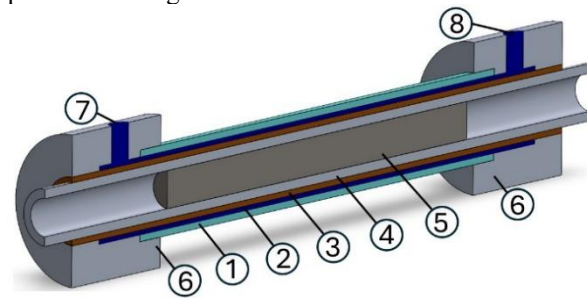


Fig. 1. Digital 3D model of the test module used in numerical simulations: 1 – glass tube, 2 – mini annular channel, 3 – copper tube, 4 – aluminium tube, 5 – cartridge heater, 6 – module head, 7 – inlet port, 8 – outlet port.

2.3 Mesh generation

A polyhedral mesh was generated using the automated meshing tools available in Simcenter STAR-CCM+ version 2020.2.1 Build 15.04.010. Local refinements were introduced near all solid–fluid interfaces and within the thermal boundary layers to ensure adequate spatial resolution of temperature and velocity gradients. The quality of the mesh was verified by analysing skewness, orthogonality, and aspect ratio. The final computational mesh contained approximately 3 969 250 cells and 14 300 508 verts, ensuring a balance between accuracy and efficiency. The boundary layers were resolved with at least five prism layers adjacent to solid surfaces. The polyhedral mesh type was selected because of its favourable numerical diffusion characteristics and stable convergence behaviour.

Figure 2 shows the 3D digital model of the test module with the generated polyhedral mesh, including a longitudinal cross-section and an enlarged view of the outlet section.

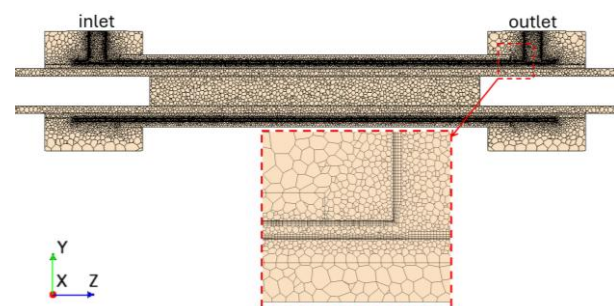


Fig. 2. Polyhedral computational mesh generated in Simcenter STAR-CCM+: longitudinal cross-section of the test module with an enlarged view of the outlet region.

2.4 Physical models and boundary conditions

The simulations assumed a single-phase flow of the working fluid. The governing equations of continuity, momentum, and energy were solved using the finite-volume method. A laminar flow model ($Re = 116$) was adopted. The boundary conditions included a prescribed mass flow rate (2.22 g/s) and an inlet temperature (294.05 K), a fixed outlet pressure (0.146 MPa), and a total heat source applied to the heater wall (48 W). The value of 48 W was selected to achieve wall temperature levels comparable to those reported in prior experimental studies while maintaining single-phase conditions, so that heat transfer is governed predominantly by forced convection rather than flow boiling.

The numerical model additionally assumed that the flow in the minichannel was incompressible with a constant mass flow rate. The material properties of the test module were considered temperature independent and heat losses to the environment were included ($12 \text{ W}/(\text{m}^2 \cdot \text{K})$) [11]. All computational domains were assigned constant density values. The basic physical properties of the materials used for the fluid and solid regions are summarised in Table 1.

Table 1. Thermophysical properties of materials used in the numerical model.

| Material | Density [kg/m ³] | Dynamic viscosity [Pa·s] | Specific heat [J/(kg·K)] | Thermal conductivity [W/(m·K)] |
|-----------|------------------------------|--------------------------|--------------------------|--------------------------------|
| Fluid | | | | |
| HFE-649 | 1600 | 0.00064 | 1103 | 0.059 |
| Solid | | | | |
| Aluminium | 2702 | - | 903 | 237 |
| Copper | 8940 | - | 386 | 398 |
| Glass | 2500 | - | 840 | 1.4 |
| Steel | 7850 | - | 486 | 52 |

2.5 Numerical procedure and convergence criteria

The simulations were performed on a workstation equipped with an Intel Core i9-10920X CPU (24 cores, 3.5 GHz) and 256 GB RAM.

The solver settings were defined to achieve stable convergence of all residuals and key monitored parameters, such as wall temperature and pressure drop. The simulations were iterated until the residuals for continuity, momentum, and energy decreased below the specified threshold values, and the calculated fields showed no further temporal variation.

Convergence was verified through monitoring of representative solution variables. The normalized residuals of continuity [12], momentum, and energy equations were reduced by at least four orders of magnitude, reaching values below 10^{-6} for momentum and energy. The monitored average wall temperature stabilised within less than 0.3% variation over 500 consecutive iterations confirming that a steady-state solution had been achieved. The final number of iterations required for full convergence was 12000.

2.6 Validation plan

The developed CFD model will be validated using data obtained from the experimental test section, which is currently being modernised at KUT. The experimental setup includes measurements of the temperature distribution along the heater wall. Thermocouples and pressure sensors are installed at the inlet and outlet of the mini annular channel to record the working fluid temperature and pressure.

The validation will involve a direct comparison between the numerical and experimental results for the following quantities: (i) axial wall temperature distribution along the selected surface of the heater and (ii) inlet/outlet fluid temperature and pressure.

Each numerical case will replicate the boundary conditions of the corresponding experiment, including the imposed heat flux, inlet temperature, outlet pressure, and mass flow rate. The relative deviation between measured and simulated quantities will be used as an accuracy metric for model validation. Validation experiments will be conducted under single-phase steady-state conditions, with measurement uncertainty analysis applied to all temperature and pressure readings.

Successful validation will confirm that the implemented numerical methodology accurately reproduces the thermo-hydraulic behaviour of the mini annular channel and can be reliably applied to parametric studies and design optimisation of compact heat exchangers.

2.7 Modification of the experimental stand

As mentioned in the *Introduction* section, previous experimental investigations at KUT were carried out using a test module with a mini annular channel. The current version of the module has been redesigned, and its view is presented in Fig. 3.

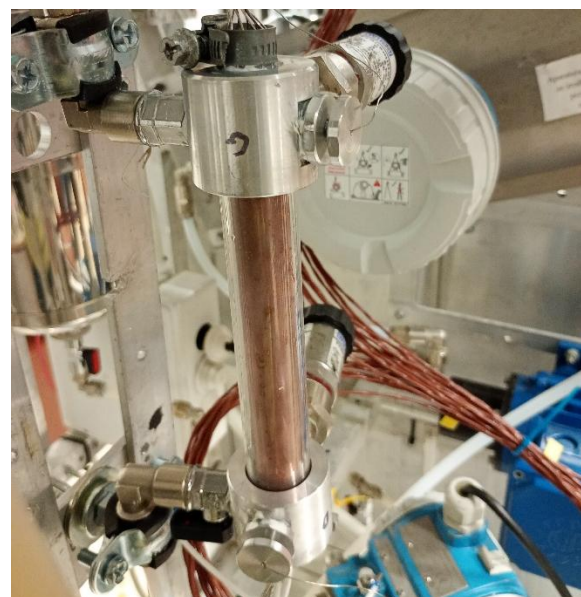


Fig. 3. View of the redesigned test module with a mini annular channel, used for ongoing experimental investigations at KUT.

The main loop of the experimental setup comprises the following components: the test module with a mini annular channel, a gear pump, a pressure regulator acting as an expansion and compensation tank, an additional heat exchanger, a Coriolis mass flow meter, an air separator, and filtration units. The system is equipped with two relative pressure transducers and thermocouples mounted at the inlet and outlet of the mini annular channel to record pressure and temperature data. The data acquisition and processing system is currently being modernised to improve the accuracy of measurement and long-term stability. A general view of the experimental stand is shown in Fig. 4.

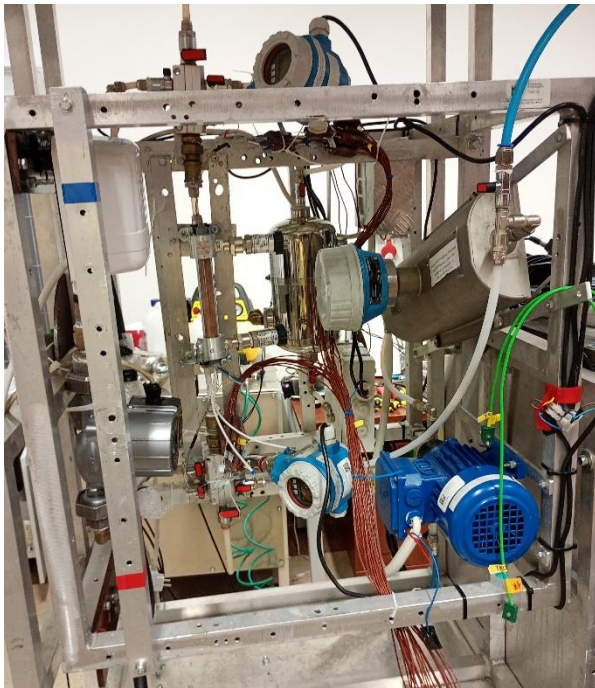


Fig. 4. General view of the experimental stand at KUT, used for investigations of heat transfer during fluid flow in minichannels.

3 Results

3.1 General outcome

Numerical simulations yielded detailed spatial distributions of the temperature, velocity, and pressure fields within the mini annular channel under steady-state conditions. These results provide quantitative information on the heat transfer and flow structure characteristics of confined laminar convection in small-scale geometries. The combination of temperature, velocity, and pressure data enables the determination of the local heat transfer coefficient based on the applied heat flux, the heated wall temperature, and the bulk fluid temperature. The resulting axial profile of the heat transfer coefficient exhibits an initial increase near the channel inlet, followed by a gradual decrease and stabilisation, corresponding to the thermal entrance and fully developed regions, respectively.

3.2 Temperature distribution

The results of the numerical simulations conducted in Simcenter STAR-CCM+ are presented in the form of a temperature distribution, as shown in Fig. 5. Furthermore, Fig. 6 illustrates the temperature dependence as a function of the distance from the inlet to the mini annular channel of the copper-fluid contact surface.

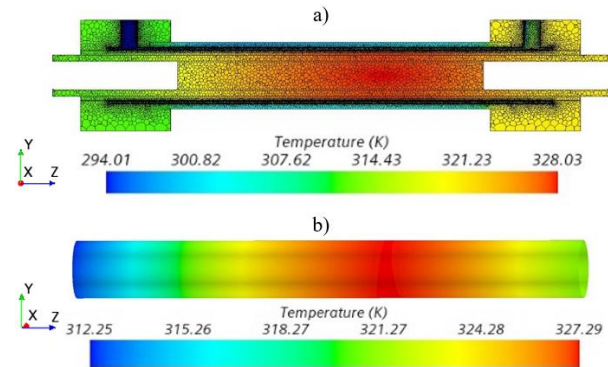


Fig. 5. Temperature distribution obtained from Simcenter STAR-CCM+ numerical results for: a) the complete test module, b) the copper-fluid contact surface; longitudinal section along the axis of symmetry of the mini annular channel.

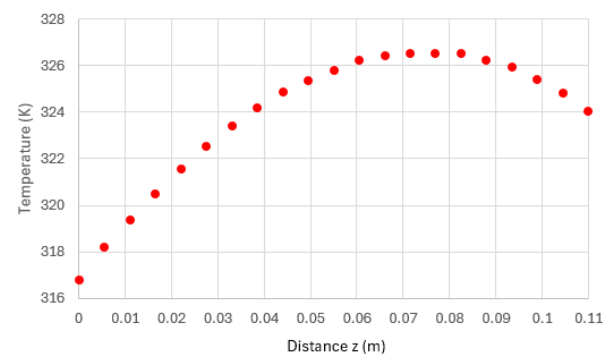


Fig. 6. Temperature dependence as a function of distance from inlet to the mini annular channel, obtained from Simcenter STAR-CCM+ numerical calculations.

When analysing the results presented in Fig. 6, it can be observed that the temperature gradually increases along the flow direction from the inlet up to approximately two-thirds of the channel length, indicating continuous heat absorption by the working fluid. In the outlet region, a slight decrease in temperature is visible, which may result from heat losses to the surroundings and the redistribution of thermal energy near the channel exit.

Temperature contours and longitudinal profiles illustrate the development of the thermal field and help to identify regions with higher temperature gradients near the inner wall. The distribution confirms a stable thermal regime with no signs of numerical instability.

3.3 Velocity field

The distribution of fluid velocity is presented in Fig. 7.

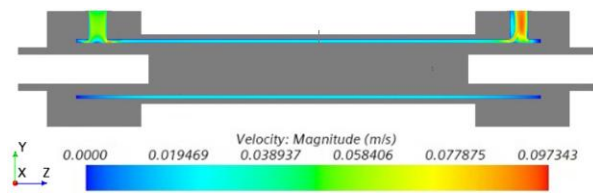


Fig. 7. Distribution of flow velocity within the fluid domain of the mini annular channel obtained from the Simcenter STAR-CCM+ simulations.

The results illustrated in Fig. 7 indicate a nearly uniform velocity profile within the main section of the mini annular channel, confirming stable laminar flow conditions. Local flow disturbances are observed near the inlet and outlet regions, where slight recirculation zones appear due to geometric transitions and boundary effects. The highest velocity values occur in the outlet section of the channel, which is consistent with the gradual acceleration of the fluid caused by thermal expansion and the imposed boundary conditions.

3.4 Pressure distribution

The distribution of the absolute pressure is shown in Fig. 8. The calculated pressure field enables the evaluation of the hydraulic characteristics of the mini annular channel. The pressure decreases monotonically along the flow path, and the overall pressure drop between the inlet and outlet is determined from the simulation results.

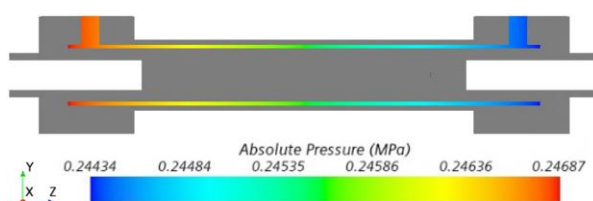


Fig. 8. Absolute pressure distribution in the fluid domain of the mini annular channel obtained from the Simcenter STAR-CCM+ numerical simulations.

3.5 Local heat transfer coefficient

The distribution of the local heat transfer coefficient in the mini annular channel, obtained from numerical simulations, is shown in Fig. 9. The coefficient was determined from the wall temperature distribution and the applied heat flux at the inner surface of the heater.

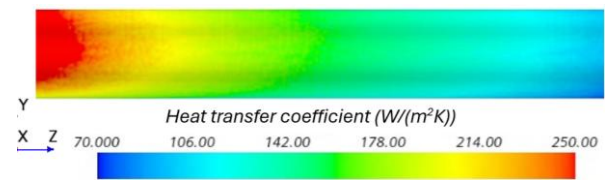


Fig. 9. Local heat transfer coefficient distribution at the copper–fluid interface of the mini annular channel obtained from Simcenter STAR-CCM+ simulations.

The results presented in Fig. 9 indicate that the local heat transfer coefficient decreases from the inlet toward the outlet of the channel. This trend corresponds to the thermal development of the flow, where the boundary layer thickens along the flow direction, reducing the local temperature gradient at the heated wall. The obtained distribution is characteristic of the transition from the thermal entrance region to the fully developed regime of laminar flow.

4 Conclusions

A numerical investigation of single-phase forced convection heat transfer in a mini annular channel was performed using the Simcenter STAR-CCM+ environment. The developed CFD model allowed for a detailed analysis of the temperature, velocity, and pressure distributions within the channel and provided the basis for determining the local heat transfer coefficients.

The results confirmed that the adopted modelling approach and boundary conditions yield physically consistent and numerically stable solutions. The simulated fields demonstrate the expected features of laminar flow: a gradual increase in the fluid temperature along the channel, a parabolic velocity profile, and a monotonic pressure drop in the flow direction. The numerical results show that the local heat transfer coefficient decreases with increasing distance from the inlet, reaching its lowest values near the outlet of the channel. This behaviour corresponds to the progressive thermal development of the flow and the thickening of the boundary layer along the heated surface.

The obtained results provide a valuable reference for ongoing experimental work aimed at validating the model under controlled laboratory conditions. Once validated, the CFD framework will serve as a predictive and optimisation tool for the design of compact heat exchangers employing mini annular channels. Future developments will include the implementation of two-phase flow analysis and conjugate heat transfer modelling to further extend the applicability of the numerical model to advanced thermal systems. The study thus forms a part of the ongoing research programme integrating CFD and experimental investigations on flow boiling and convective heat transfer in mini-scale geometries.

Acknowledgments

This research was funded in part by the National Science Centre, Poland, grant no. UMO-2025/57/B/ST8/00907.

For the purpose of Open Access, the authors have applied a CC BY public copyright licence to the Author Accepted Manuscript (AAM) version arising from this submission.

The calculations were made using the Simcenter Star-CCM+ software developed by Siemens PLM Software Inc., provided by GMSysystem which is a supplier of Siemens Digital Industries Software in Poland.

References

1. J. Dirker, J. P. Meyer, Heat Transfer Coefficients in Concentric Annuli, *J. Heat Transfer* **124**(6), 1200-1203 (2002).
<https://doi.org/10.1115/1.1517266>
2. G. Lu, J. Wang, Experimental investigation on heat transfer characteristics of water flow in a narrow annulus, *Appl. Therm. Eng.* **28**(1), 8-13 (2008).
<https://doi.org/10.1016/j.applthermaleng.2007.03.019>
3. S. A. Jayarajan, U. Azimov, CFD Modeling and Thermal Analysis of a Cold Plate Design with a Zig-Zag Serpentine Flow Pattern for Li-Ion Batteries, *Energies* **16**(14), 5243 (2023).
<https://doi.org/10.3390/en16145243>
4. V. Y. Agbodemegbe, X. Cheng, E. H. Akaho, F. K. Allotey, Correlation for cross-flow resistance coefficient using STAR-CCM+ simulation data for flow of water through rod bundle supported by spacer grid with split-type mixing vane, *Nucl. Eng. Des.* **285**(15), 134-149 (2015).
<https://doi.org/10.1016/j.nucengdes.2015.01.003>
5. S. Kastrati, T. Hyhlik, R. Kalinay, Modelling of Evaporation from Thin Horizontal Liquid Film, in *Topical Problems of Fluid Mechanics 2018* (Institute of Thermomechanics, AS CR, v.v.i., 2018), 167–174.
6. B. Maciejewska, M. Piasecka, A. Piasecki, The Study of the Onset of Flow Boiling in Minichannels: Time-Dependent Heat Transfer Results, *Heat Transf. Eng.* **43**(3-5), 223-237 (2021).
<https://doi.org/10.1080/01457632.2021.1874181>
7. M. Piasecka, T. Musiał, A. Piasecki, Cooling liquid flow boiling heat transfer in an annular minigap with an enhanced wall, *EPJ Web Conf.* **213**, 02066 (2019).
<https://doi.org/10.1051/epjconf/201921302066>
8. B. Maciejewska, P. Łabędzki, A. Piasecki, M. Piasecka, Comparison of FEM calculated heat transfer coefficient in a minichannel using two approaches: Trefftz base functions and ADINA software, *EPJ Web Conf.* **143**, 02070 (2017).
<https://doi.org/10.1051/epjconf/201714302070>
9. A. Pawińska, A. Piasecki, N. Dadas, S. Hożejowska, M. Piasecka, Homotopy Perturbation Method with Trefftz Functions and Simcenter STAR-CCM+ Used for the Analysis of Flow Boiling Heat Transfer, *Acta Mech. Autom.* **18**(2), 233-243 (2024).
<https://doi.org/10.2478/ama-2024-0027>
10. R. Eymard, R. Gallouët, T. Herbin, Finite Volume Methods, in *Handbook of Numerical Analysis, Vol. 7: Solution of Equation in R (Part 3), Techniques of Scientific Computing (Part 3)*, (Elsevier, Amsterdam, 2000)
11. H. Recknagel, E. Sprenger, E.-R. Hönnmann, H.-J. Schramek, *Taschenbuch für Heizung + Klimatechnik [Handbook for Heating and Air Conditioning]*, 66th ed., ISBN: 978-3486262124, (Oldenbourg Verlag, Munich, 1991)
12. Siemens PLM Software Inc., *Simcenter STAR-CCM+ Documentation* (Siemens PLM Software Inc., Plano, TX, USA, 2020)