Proceedings of the 10th International Conference on Fluid Flow, Heat and Mass Transfer (FFHMT'23)
Ottawa, Canada – June 07-09, 2023
Paper No. 181
DOI: 10.11159/ffhmt23.181

Hypervapotron High Heat Flux Cooling Numerical and Experimental Study

Vojtech Smolik[1,2], Miroslav Gleitz[2], Slavomir Entler[1,2], Pavel Zacha[2]
[1]Institute of Plasma Physics of the Czech Academy of Sciences
U Slovanky 2525/1a, Prague, Czech Republic
vojtech.smolik@fs.cvut.cz; entler@ipp.cas.cz
[2]Czech Technical University in Prague, Faculty of Mechanical Engineering
Technická 4, Prague, Czech Republic
miroslav.gleitz@fs.cvut.cz, pavel.zacha@fs.cvut.cz

Abstract **-** High heat flux cooling is one of the leading engineering challenges of the nuclear fusion reactor construction. Plasma facing components (divertor targets and the first wall) of tokamaks are operating under extreme heat load conditions. ITER first wall hypervapotron cooling channel is designed to withstand high heat fluxes up to 7 MW/m2. Water cooled hypervapotron is investigated by CFD simulation in ANSYS Fluent and the results are experimentally validated. Numerical solutions of various CFD codes are also compared to evaluate the ability of each numerical approach to solve subcooled boiling regime.

Keywords**:** hypervapotron, nuclear fusion, tokamak, subcooled boiling, ANSYS Fluent

1. Introduction

Subcooled boiling represents an important physical phenomenon for high heat flux cooling. The hypervapotron geometry creates conditions to enhance the nucleate boiling heat removal. Hypervapotron is defined by cooling fins perpendicular to the flow of cooling liquid. The central coolant stream in the channel creates a secondary vortex flow between fins. Evaporating vapour bubbles are detached from heated wall by the secondary vortex flow and diverted to central stream of water, thus maintaining the nucleate boiling regime, and enhancing the total heat flux removal [1][2].

The Box Scraper 3x4 hypervapotron geometry (Fig. 1) is examined in this article. This geometry corresponds with the hypervapotron installed on tokamak JET in Culham in the United Kingdom. Hypervapotrons in JET are being used for example as beam stopping elements in both Neutral Beam Injectors and in the Neutral Beam Test Bed. ANSYS Fluent numerical results of hypervapotron CFD analysis are validated by our experimental device HEFEL (Heat Flux Experimental Loop) on lower heat flux parameters. Experimental data for higher heat flux validation are obtained from other researchers [3]. The results of ANSYS Fluent 23.1 simulation are compared with various CFD codes (ANSYS Fluent 16.2, ANSYS CFX, STAR CCM+).

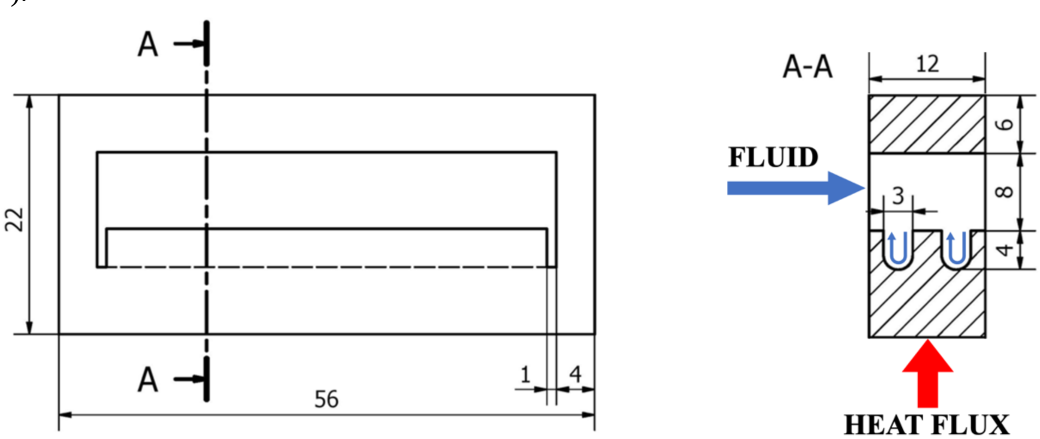

Fig. 1: Hypervapotron (Box Scraper 3x4) geometry.

2. Methods

The experimental results measured on HEFEL device are used for validation of numerical results with variable water mass flow. The temperature profile of hypervapotron solid body is compared with thermometer measurements. The induction source used for hypervapotron sample heating is controlled manually, thus the variable heat flux experiment on HEFEL will result in a significant measurement inaccuracy. Water mass flow represents a suitable variable for experimental measurement, as it can be fully regulated and measured by the control system.

Data from other researchers [3] are presented to validate numerical models for high heat fluxes, unachievable on the HEFEL device. Experimental data with variable heat flux up to 12 MW/m^2, 50 °C inlet temperature and 6 bar pressure ensures the nucleate boiling regime can be fully developed in the measured range. A selection of various CFD codes is compared on the same case boundary conditions to evaluate its ability to simulate the subcooled boiling regime.

2.1. Experimental Setup

The schematic of HEFEL device is shown in the Fig. 2. Circulating water is regulated by control system to a stable inlet temperature. Total pipe length of the experimental loop is 10 m, maximal coolant pressure and temperature is 10 bar and 110 °C respectively. The hypervapotron test sample (Fig. 2) is installed in the test section of experimental loop. The temperature and pressure difference of test section is measured to evaluate the pressure drop of experimental sample and the heat transfer calorimetry. Flowmeter provides mass flow measurements. Heating of the hypervapotron sample is provided by induction source with 15 kW input power. CuCrZr hypervapotron body is equipped with steel sheets on two sides to enhance the induction heating power. 2 mm steel sheets have been numerically confirmed in ANSYS Maxwell 3D as optimal geometry to achieve higher heating power compared to solid CuCrZr. This material modification allows to achieve heating power up to 3.5 kW resulting in hypervapotron heat flux up to 5 MW/m^2 [8].

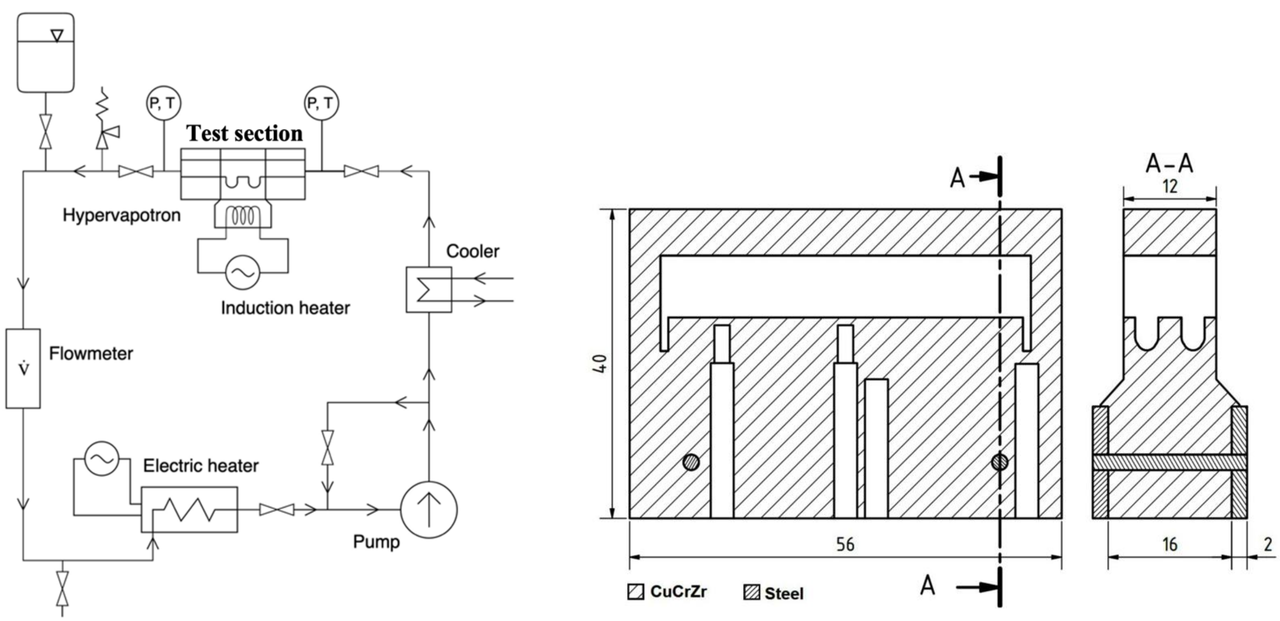

Fig. 2: HEFEL experimental loop (left) and the hypervapotron test sample (right).

The Figure 2 (right) also shows the position of gaps for temperature measurement in hypervapotron test sample body. The platinum Pt100 resistance thermometers inside the solid body are used for numerical model validation. The gaps for resistance thermometers are 3 mm in diameter. The thermometer gaps are included both in the experimental and numerical model. The future hypervapotron samples will include 1 mm diameter thermocouples to reduce impact of the gaps on the temperature profile measurement.

Hypervapotron test sample is installed in the body of test section channel (Fig. 3), surrounding walls are insulated to prevent heat loss by conduction. There is a total of 8 thermometers installed in the test sample, thermometers positions are shown in the Figure 3 (right). Experimental temperature measurements are compared with CFD temperature profile.

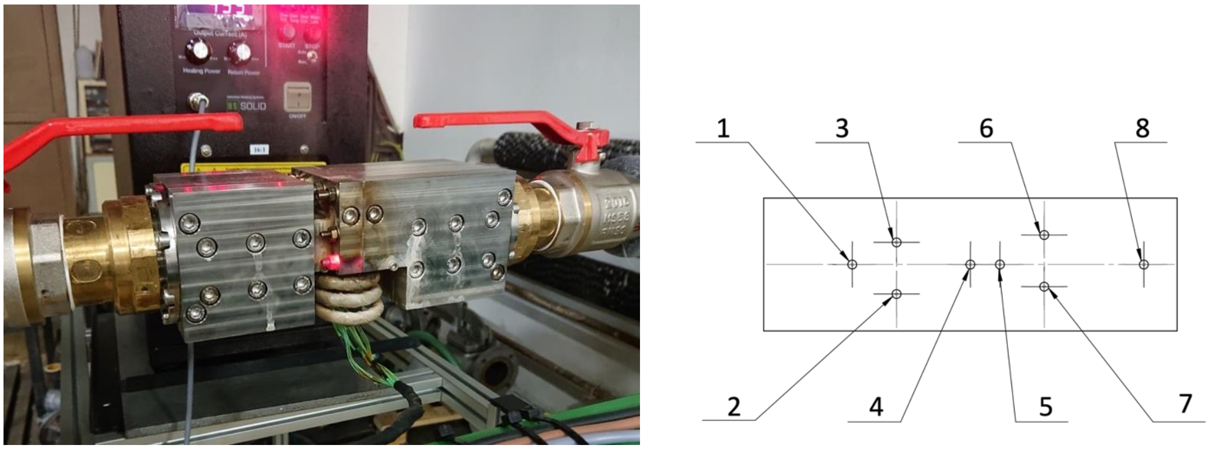

Fig. 3: Test section of HEFEL loop in operation (left) and the hypervapotron test sample thermometer positions (right).

2.2. Numerical model

Numerical simulation has been performed in ANSYS Fluent 23.1. One half symmetry of hypervapotron geometry has been considered to reduce the number of mesh cells and computational time. Tetrahedral type mesh with inflation layers on interface walls consists of 479 783 cells. Table 1 describes used numerical models. Eulerian multiphase model allows to simulate the behaviour of subcooled boiling vapour bubbles. Boiling model and two-phase flow parameters are based on previous works by Milnes [4] and Písek [5]. The frequency of bubble departure is defined by user function, it is set to a constant value of 412 Hz. This frequency corresponds with the vortex rotation velocity between hypervapotron fins, simplifying the complex physical phenomenon simulation.

Table 1: ANSYS Fluent 23.1 numerical model setup.

Solid material	CuCrZr
Fluid material	Water
Viscous model	k-epsilon
K-epsilon model	Realizable
Near-wall treatment	Enhanced wall treatment
Multiphase model	Eulerian
Boiling model	RPI
Volume fraction	Implicit

Bubble departure diameter	Tolubinski Kostanchuk
Frequency of bubble departure	UDF, 412 Hz
Nucleation site density	Lemmert Chawla
Area influence coefficient	Delvalle Kenning
Drag coefficient	Schiller Naumann
Lift coefficient	Tomiyama
Turbulence interaction	Troshko Hassan
Surface tension coefficient	0.046845 N/m

3. Results

The experimental results measured on HEFEL device are used for validation of numerical results with variable water mass flow. The induction source used for hypervapotron sample heating is controlled manually, thus the variable heat flux experiment on HEFEL will result in a significant measurement inaccuracy. Water mass flow represents a suitable variable for experimental measurement, as it can be fully regulated and measured by the control system.

Data from other researchers [3] are presented to validate numerical models for high heat fluxes, unachievable on the HEFEL device. Experimental data with variable heat flux up to 12 MW/m^2, 50 °C inlet temperature and 6 bar pressure ensures the nucleate boiling regime can be fully developed in the measured range.

3.1 Variable inlet mass flux (velocity)

The experimental results of HEFEL measurement are shown in the Figure 4. The induction heating source has been set to provide a stable 5 MW/m^2 heat flux. Thermometer T2, T3 and pressure drop experimental measurements are plotted in relation to the variable mass flow (0.367 kg/s = 1 m/s). A full range of experimental results is plotted, an average value for each datapoint is highlighted. The uncertainties of resistance thermometer measurements ranges from ± 1.6 % to ± 4.3 %. The arithmetic mean inlet water temperature is 80.01°C with a 0.21 °C standard deviation. The results of ANSYS Fluent 23.1 numerical analysis are represented by a dotted line.

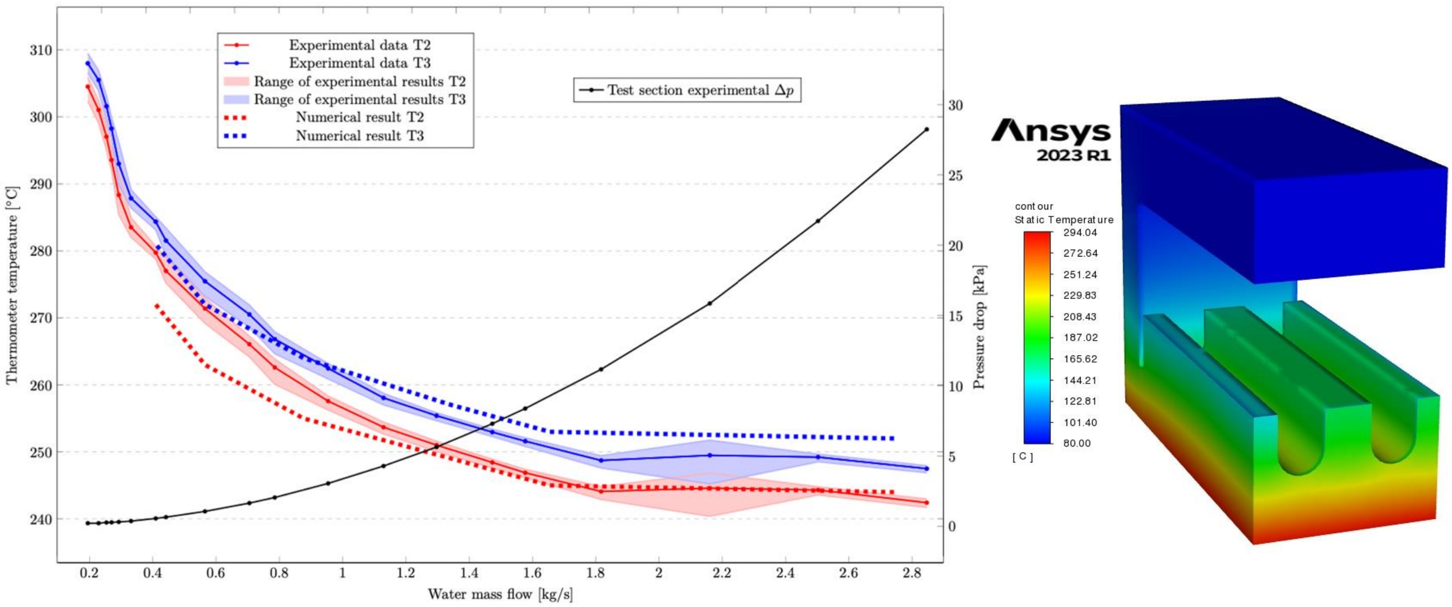

Fig. 4: Experimental and numerical results, variable mass flow, inlet: 80 °C, 2 bar, 5 MW/m^2 and temperature profile (2.5 kg/s).

3.2. Variable heat flux

Experimental data for CFD model validation are obtained from *Design issues and fatigue lifetime of hypervapotron elements of the JET neutral beam injectors* [3]. Hypervapotron channel samples were tested using electron beams, reaching heat flux up to 10 MW/m^2. Inlet fluid parameters are 50 °C and 4 m/s. The operating pressure is set to 6 bar, increasing the boiling point of water to 159 °C. The experimental data are compared with numerical analysis by previous researchers solved in ANSYS CFX 2010 (Milnes) [4], ANSYS Fluent 2016 (Písek) [5], ANSYS CFX 2017 (Pitoňák) [6], STAR CCM+ 2022 (Gleitz) [7] and ANSYS Fluent 2023 (Smolík).

The temperature profile and vapour volume fraction distribution at 4 MW/m^2 heat flux is shown in the Figure 6. The nucleate boiling is beginning to appear in the hottest region between hypervapotron fins, near the symmetry plane of the sample. The upper region of the hypervapotron solid body maintains the same temperature as the inlet water.

The Figure 7 shows results of numerical analysis. Thermocouple temperature rise is related to the difference between thermocouple measurement in solid body and 50 °C fluid inlet temperature. Numerical codes are compared with experimental data. The nucleate boiling regime is expected to be fully developed at heat flux around 4 MW/m^2, to interpret the drop of temperature gradient in the experimental data. For heat fluxes over 10 MW/m^2, numerical and experimental results suggest the departure from nucleate boiling between the hypervapotron fins. An insulating vapour layer between fins decreases the heat transfer coefficient, thus the solid temperature is rapidly rising.

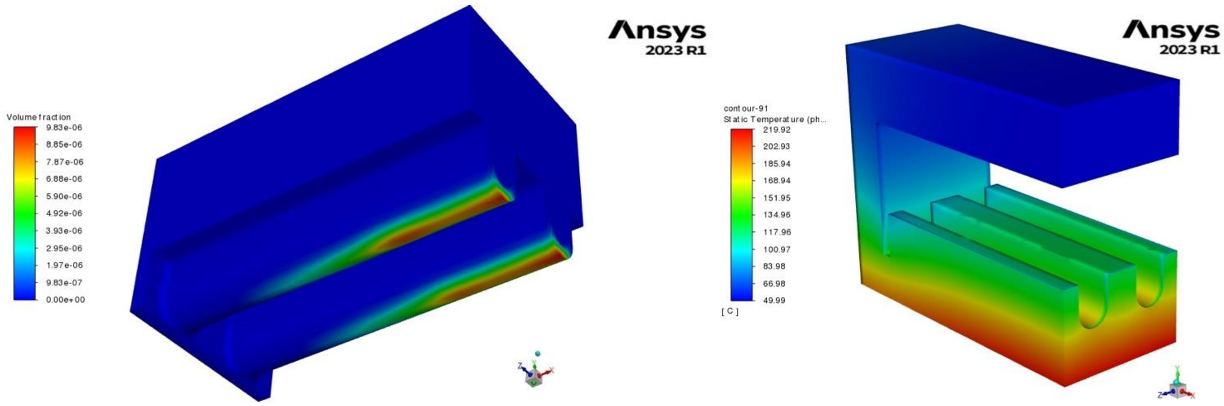

Fig. 6: Numerical results: vapour volume fraction (left) and temperature profile (right), 4 MW/m^2, inlet water: 50 °C, 6 bar, 4 m/s.

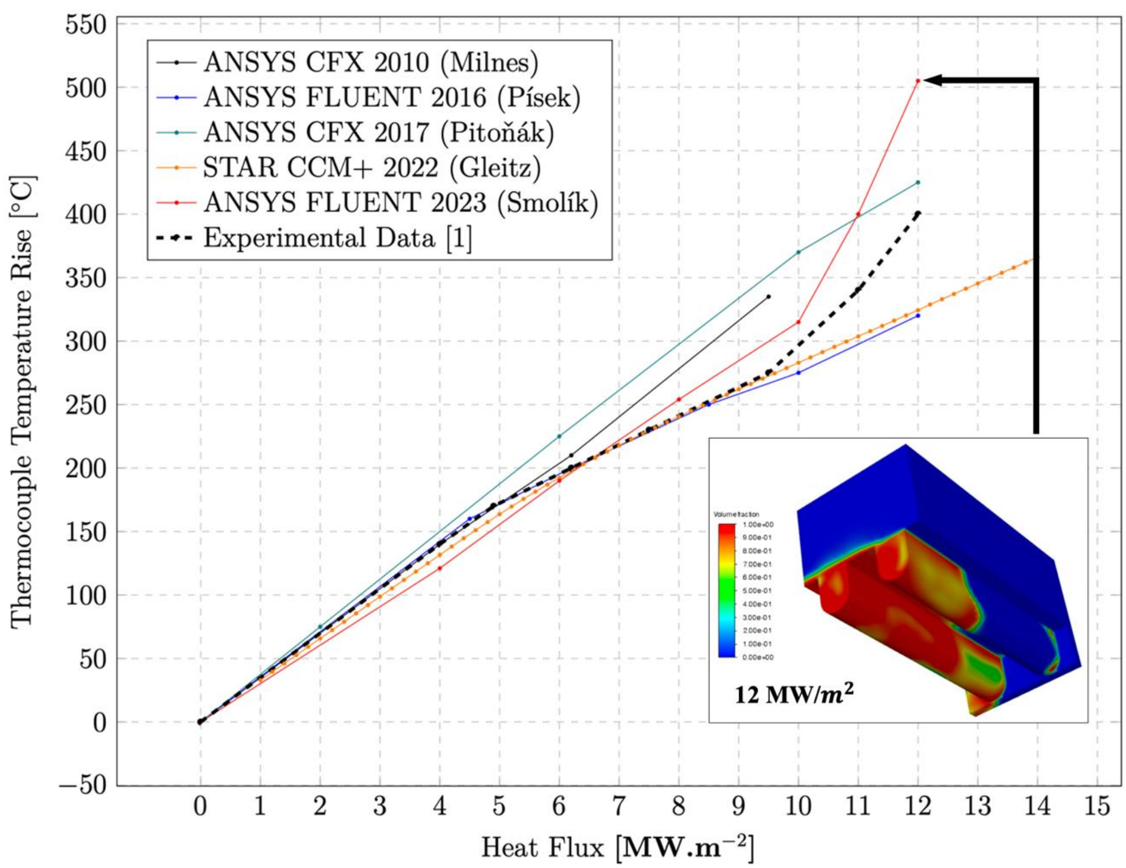

Fig. 7: Experimental and numerical results, variable heat flux, inlet water: 50 °C, 6 bar, 4 m/s.

4. Discussion

Numerical results perform a well agreement with experimental measurements on high water mass flow over 1 kg/s (thermometer T2). There is observable discrepancy (up to 10 °C) between numerical and experimental data on lower mass flow. The discrepancies at lower mass fluxes may be related to the nucleate boiling regime formation in this area. Based on this experimental measurement, the numerical model is considered valid in the high fluid velocity range in the position of thermometer T2.

Numerical simulations by Milnes and Pitoňák (ANSYS CFX) are based on the k-omega SST viscous model, while Smolík and Písek (ANSYS Fluent) analysis uses the k-epsilon viscous model. None of the numerical analysis results clearly indicates the presence of all hypervapotron boiling regimes observable by “S” shaped temperature curve in Figure 7 experimental results. However, ANSYS Fluent 23.1 simulation successfully predicted the departure from nucleate boiling and rapid temperature increase in the solid body at high heat fluxes. This aspect represents a compeling development of ANSYS Fluent abilities between version 16.2 and 23.1, as this observation is not reported by Písek in his analysis.

5. Conclusion

Subcooled boiling regime in hypervapotron is a complex thermohydraulical phenomenon, a severe challenge for CFD simulation. The development of numerical models is limited by a lack of relevant experimental data for validation. The HEFEL experimental device has a limited induction heating source, available to achieve a 5 MW/m^2 maximal heat flux. Further experimental research including the laser beam heat flux source is in preparation at the present time.

Experimental measurement on HEFEL device has been performed to validate the ANSYS Fluent 23.1 numerical simulation. Experimental data of 5 MW/m^2 heat flux and variable mass flow rate successfully validated the numerical analysis results in the range of high fluid velocities (single-phase/very low vapour volume fractions). The behaviour of vapour bubbles at lower fluid velocities caused an observable difference between numerical and experimental results

Comparison of numerical results with experimental data from literature is presented to evaluate the CFD model performance on high heat fluxes up to 12 MW/m^2 (unachievable on the HEFEL device). None of the numerical analysis is considered valid in a full range of heat fluxes. However, ANSYS Fluent 2023 solution successfully simulated the formation of insulating layer of vapour between hypervapotron fins, reducing the heat transfer coefficient and consequently increasing the solid body temperature.

Acknowledgements

Acknowledgement: The work was supported from the European Regional Development Fund – Project ”CAAS” (No.CZ.02.1.01/0.0/0.0/16-019/0000778) , and by the Strategy AV21 of the Czech Academy of Sciences within the research program “Sustainable Energy”.

References

[1] Y. Wang and A. Dedov, “An review for the heat transfer researches of hypervapotron in the iter first wall,” Journal of Physics: Conference Series, vol. 1683, p. 022100, 12 2020.

[2] G. Dell’Orco, W. Curd, J. Berry, K. Chang, J. Ferrada, B. Gopalapillai, D. Gupta, S. Kim, I. Kuehn, A. Kumar, F. Li, A. Petrov, and W. Reiersen, “Design modification and optimization of the iter cooling water system,” Fusion Engineering and Design, vol. 86, no. 1, pp. 15–19, 2011.

[3] D. Ciric, M. Akiba, H.-D. Falter, D. Martin, K. Sato, and K. Yokoyama, “Design issues and fatigue lifetime of hypervapotron elements of the jet neutral beam injectors” in *18th IEEE/NPSS Symposium on Fusion Engineering. Symposium Proceedings* (Cat. No.99CH37050), pp. 407–410, 1999.

[4] J. Milnes, Computational Modelling of the HyperVapotron Cooling Technique for Nuclear Fusion Applications. PhD thesis, Department of Aerospace Sciences Cranfield University, Cranfield, UK, 2010.

[5] V. Písek, “Ověření aplikovatelnosti cfd metodik pro výpočty podchlazeného varu,” Master’s thesis, ČESKÉ VYSOKÉ UČENÍ TECHNICKÉ V PRAZE, Fakulta Strojní, Prague, 2016.

[6] V. Pitoňák, “Studie podchlazeného varu v hypervapotronu,” Master’s thesis, ČESKÉ VYSOKÉ UČENÍ TECHNICKÉ V PRAZE, Fakulta Strojní, Prague, 2017.

[7] M. Gleitz, P. Zacha, S. Entler, and J. Syblík, “Design of experimental device for testing of subcooled flow boilling,”, Engineering Mechanics 2020, pp. 150–153, 01 2020.

[8] M. Gleitz, V. Smolík, P. Zacha and S. Entler, “Commissioning of the high heat flux experimental loop”, Symposium on Fusion Technology 2022, poster n. 166, 2022.