MIGRATE2018:210642

COMPUTATIONAL STUDY AND EXPERIMENTAL VALIDATION OF MICRO HEAT EXCHANGER PERTURBATORS FOR HIGH-TEMPERATURE MICRO-CHP APPLICATIONS

Jojomon Joseph[a,b,*], Rabia Nacereddine[a], Michel Delanaye[a], Jan G. Korvink[b], Juergen J. Brandner[b]

[a]Mitis, Liege, Belgium

[b]IMVT, Karlsruhe Institute of Technology, Campus North, Eggenstein-Leopoldshafen, Germany

KEY WORDS

Reduced model, Experimental validation, S-shaped and wire net protrusions, collector maldistribution

Abstract

The objective of this paper is to predict compact heat exchanger performance by a detailed modelling of complex microchannels (with wire-net and S-shaped perturbators) for a miniaturised Combined Heat and Power (CHP) system. A new CFD methodology was developed [1, 2] and experimentally validated in this paper to assess the collector performance based on reduced order modelling. In this paper, we also compare the microchannel as well as the collector performances of two different models: Wire-net model and the S-shaped model. The wire-net heat exchanger (Figure 1 (a, b, c)) model is assembled as a stack of counter-flow passages with optimised thickness separated by thin foils [1, 2, 3]. A metallic wire mesh is inserted in the flow passages (Figure 1 (b)) to provide the required stiffness and enhance the microchannel efficiency (localised turbulence). The S-shaped heat exchanger model consists of microchannels with S-shaped protrusions (Figure 2 (a)) along the counter-flow passages with an integrated collector (Figure 2 (b)).

A microchannel parametric study was conducted on various heat exchanger parameters to optimise both models for the micro-CHP system. The Reynolds number based on the hydraulic diameter of the inlet is between 30-300 (for micro-channels), 500-30000 (for secondary collectors) and even higher for primary collectors. Wire-net microchannel porosity (78-82%) can enhance localized turbulence at smaller Reynolds numbers. Since the Knudsen number is low, thermal creep or rarefaction effects are negligible. The thermal efficiency, ε (%) is calculated using the relation,

$$\varepsilon, \% \; = \frac{C_h(T_{in}^{h} - T_{out}^{h}\}}{\min(\{C_{cold}, C_{hot}\})\,(T_{in}^{hot} - T_{in}^{c}\}} * 100 \qquad 1$$

$$\mathrm{C}_{\{\mathrm{cold}\}} = \mathrm{m}_{\{\mathrm{cold}\}}\mathrm{Cp}_{\{\mathrm{cold}\}};\, \mathrm{C}_{\{\mathrm{hot}\}} = \mathrm{m}_{\{\mathrm{hot}\}} * \mathrm{Cp}_{\{\mathrm{hot}\}}$$

The total pressure drop, ΔP, % is calculated using,

$$\Delta P, \% \; = \frac{\{\, P_{in} - \; P_{out}\}}{\{P_{ab}\}} * 100 \qquad 3$$

*joseph.jojomon@mitis.be

From the Conjugate Heat Transfer (CHT) microchannel analysis of both models, there is an optimum mass flow where the microchannel thermal efficiency (Eq. (1)) reaches its maximum (See Figure 1 (d)). However, a substantial decrease in efficiency at higher mass flow rates has been encountered [1, 2, 3]. The steepness of this efficiency curve pattern is stronger for the wire net design compared to the S-shaped microchannels. However, for the wire-net microchannels (specially at Reynolds numbers higher than the optimum mass flow) the pressure loss (Eq. (3)) is relatively high together with a lower thermal efficiency in comparison with S-shape perturbators. These performance characteristics were utilised to reduce the size and cost of the preliminary heat exchanger design.

The developed CFD methodology(which comprises CHT analysis and Reduced order modelling, See Figure 3) is to predict the collector performance with good accuracy and reduce the computational grid size to a considerable extent. The Reduced Model (RM) is implemented to replicate the microchannel pressure losses and a source term implementation to bring the temperature effects. The Reduced Model consists of collectors and microchannels, modelled through a porous medium approximation [4]. The porous medium model [5, 6], based on the Darcy-Forchheimer law, is modified (Constant Integration Method) to account for the high-temperature evolution in the heat exchanger. The modified equation takes into account the high temperature variation inside the microchannels along with the localized turbulence effects [7]. The resulting microchannel characteristics (pressure drop, temperature drop etc.) from a series of 3D-CFD Conjugate Heat Transfer analysis using Ansys Fluent is used to calculate the inertial and viscous coefficients using the Constant Integration method. These characteristics have been implemented and numerically verified. The best revised methodology allows obtaining pressure drop with less than three percent error with respect to the full 3D-CFD Conjugate Heat Transfer modelling for Brayton and Inverted Brayton micro-CHP cycles (See Figure 4) [2].

A detailed CHT analysis of the microchannels with S-shaped perturbators showed that the overall heat transfer coefficient is directly correlated with the Turbulent Reynolds stress term, uv. This is induced by the strong recirculation zones induced by the S-shaped structures. The turbulence production term, uv·Du/DY is higher for a particular S-shape arrangement. The increased wire net pressure loss could not enhance the thermal efficiency like the S-shaped perturbators. Linearly arranged S-shaped perturbators with an optimum spacing (See Figure 2 (a)) found to be the best among all other arrangements. The collector design and optimization was found to be a major challenge due to its complex outlets and inlet design (See Figure 2 (b)).

Besides, a comprehensive investigation of the microchannel wire net flow physics is made using a higher order Reynolds Stress turbulence model to obtain the full velocity gradient tensor. This could investigate the effect of anisotropic flow physics in the isotropic wire net microchannels. Lambda 2 criteria was implemented to investigate the flow mixing of the centrally convected non-disturbed mass flow. Furthermore, the analysis of the turbulence production terms provided a deeper insight into flow attachment and detachment near the wire net intersections. Localised turbulence can enhance the heat exchanger performance. Besides, this will also increase the pressure losses too. Shifting the Reynolds critical to smaller Reynolds number by using perturbators will control the pressure losses and enhance the thermal efficiency to a considerable extend.

The experimental test bench configuration is detailed in Figure 5. To reduce heat losses to the environment, there is a 5 cm layer of insulating material all around the heat exchanger to

*joseph.jojomon@mitis.be

replicate the adiabatic wall boundary conditions of the Reduced Model. Furthermore, pressure sensors are installed in the primary collectors that feed the mass flow to the secondary collectors and then to the microchannels. The experimental validation showed that the reduced model predicts the overall heat exchanger performance with a reasonable good accuracy (See Figure 6).

Acknowledgement

This project MIGRATE is funded by the European Union's Horizon 2020 research and innovation program under the Marie Sklodowska-Curie grant agreement No. 643095.

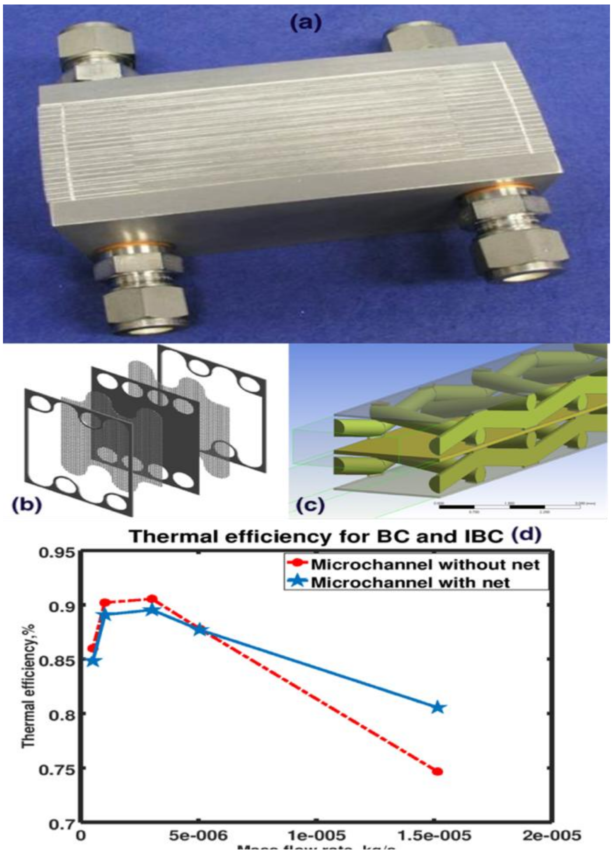


Figure 1. (a) Wire net brazed heat exchanger, (b) Arrangement, (c) Numerical Conjugate Heat Transfer domain, (d) Thermal efficiency curve for various mass flows.

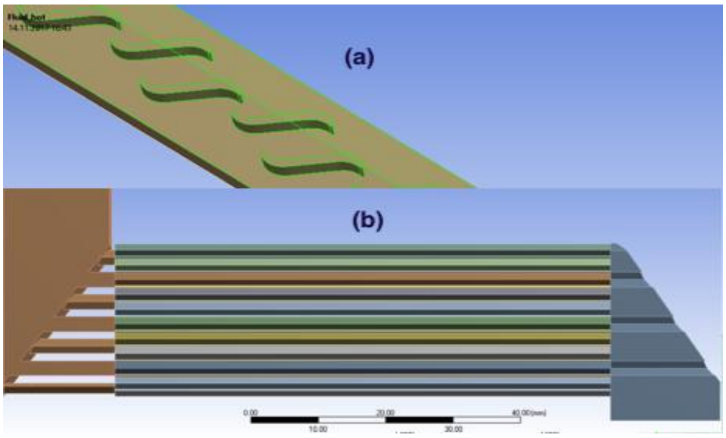
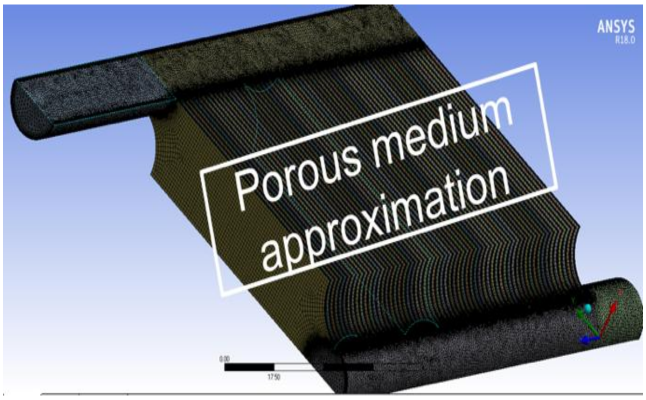


Figure 2. S-shaped perturbators, (a) Microchannel design, (b) Collector design

*joseph.jojomon@mitis.be



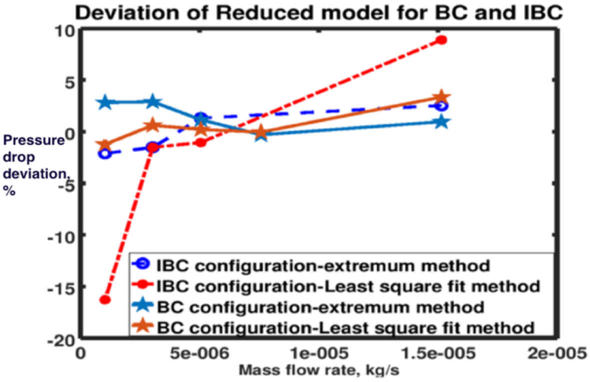
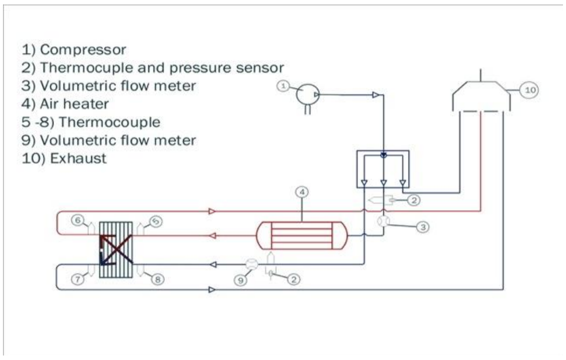
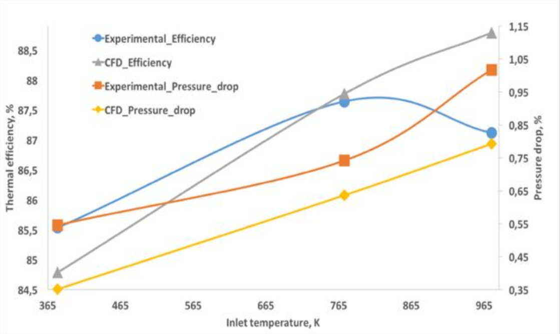


Figure 3. Reduced Model (mesh) to investigate collector performance **Figure 4. Numerical validation for RM**





Fig

References

[1] J. Joseph *et al.*, "Design & optimization of a compact heat exchanger for a micro gas turbine micro-chp application", presented at the First Migrate Summer School and Workshop, June 27 to July 1, 2016, Strasbourg, France.

[2] J. Joseph *et al.*, "Advanced CFD methodology to analyze high-temperature wire net compact heat exchangers for a micro-CHP application", presented at the Second Migrate Summer School and Workshop, June 26 to 30, 2017, Sofia, Bulgaria.

[3] M. Delanaye *et al.*, "Development of a recuperated flameless combustor for an inverted Bryton cycle microturbine used in residential micro-CHP", presented at the Proceedings of the ASME 2017 Turbomachinery Technical Conference and Exposition, TurboExpo Charlotte, USA, June 26–30, 2017.

[4] Yizhou, Yan, Rizwan-uddin, & Nahil, Sobh (2005). "CFD simulation of a research reactor." presented at the Int. topical meeting on mathematics and computation, supercomputing, reactor physics and nuclear and biological applications, France: SFEN, 2005.

[5] D.A. Nield, A. Bejan, (2013). Convection in Porous Media, 4th edition (Springer Inc., New York, 2013).

[6] Nicolas G. (2011). High temperature and pressure reactive flows through porous media. International Journal of Multiphase Flow, Elsevier, 2011, 37 (1), pp.24-35.

[7] Peter S. Bernard & James M. Wallace (2002). Turbulent Flow: Analysis, Measurement & Prediction. John Wiley & Sons. p. 324. ISBN 0471332194.1.

[8] J. Joseph *et al.*, "Advanced CFD methodology to investigate high-temperature complex wire-net micro heat exchange performance.", presented at the MicroFluidics and Non-Equilibrium Gas Flows Conference 2018, Feb. 28 to March 2, 2018, Strasbourg, France.

*joseph.jojomon@mitis.be