ADVANCED CFD METHODOLOGY TO INVESTIGATE HIGH- TEMPERATURE COMPLEX WIRE NET MICRO HEAT EXCHANGER PERFORMANCE

Jojomon Joseph, Michel Delanaye, Rabia Nacereddine, Juergen Brandner

▶ To cite this version:

Jojomon Joseph, Michel Delanaye, Rabia Nacereddine, Juergen Brandner. ADVANCED CFD METHODOLOGY TO INVESTIGATE HIGH- TEMPERATURE COMPLEX WIRE NET MICRO HEAT EXCHANGER PERFORMANCE. Proceedings of the 5th European Conference on Microfluidics – μFlu18 3rd European Conference on Non-Equilibrium Gas Flows – NEGF18, Feb 2018, Strasbourg, France. hal-02406575

HAL Id: hal-02406575
https://hal.science/hal-02406575

Submitted on 12 Dec 2019

µFLU-NEGF18-45

ADVANCED CFD METHODOLOGY TO INVESTIGATE HIGH-TEMPERATURE COMPLEX WIRE NET MICRO HEAT EXCHANGER PERFORMANCE

Jojomon Joseph[1] , Michel Delanaye[*1], Rabia Nacereddine[2], Juergen J. Brandner[3]
[1] joseph.jojomon@mitis.be , [*1] michel.delanaye@mitis.be , [2] rabia.nacereddine@mitis.be ,
[3]juergen.brandner@kit.edu.

KEY WORDS

Conjugate heat transfer, Micro channels, Counter flow, Wire net heat exchanger.

SHORT SUMMARY

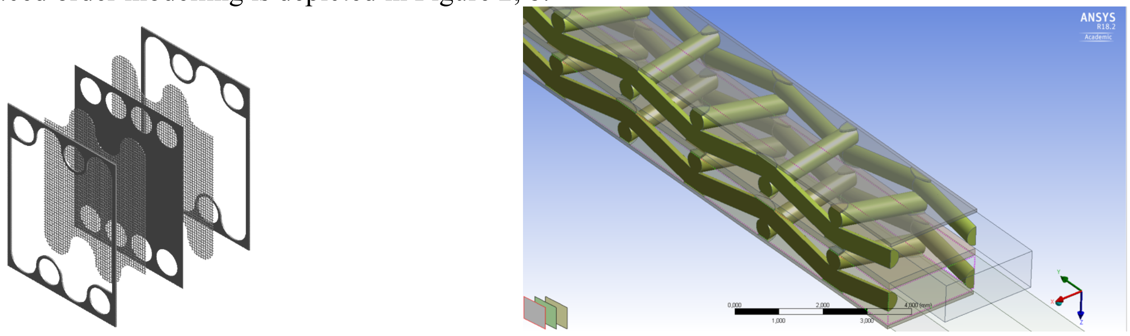
The objective of this paper is to predict micro heat exchanger performance for a micro Combined Heat and Power systems by a detailed modelling of complex microchannels and a new CFD methodology to assess the entire heat exchanger characteristics based on reduced order modelling. CFD methodology comprises of Conjugate Heat Transfer models and Reduced order models. This could reduce the computational size to a considerable large extent (billion cells to few million cells) with good accuracy. The porous medium model, based on Darcy-Forchheimer law is modified (Constant Integration Method) to account for the temperature evolution. They have been implemented and verified. The best-revised methodology allows obtaining pressure losses with less than three percent error with respect to the 3D CFD-CHT modelling. Higher order turbulence models were used to investigate the influence of mesh on heat exchanger performance. A parametric study was conducted to study the influence of microchannel parameters on heat exchanger performance.

EXTENDED ABSTRACT

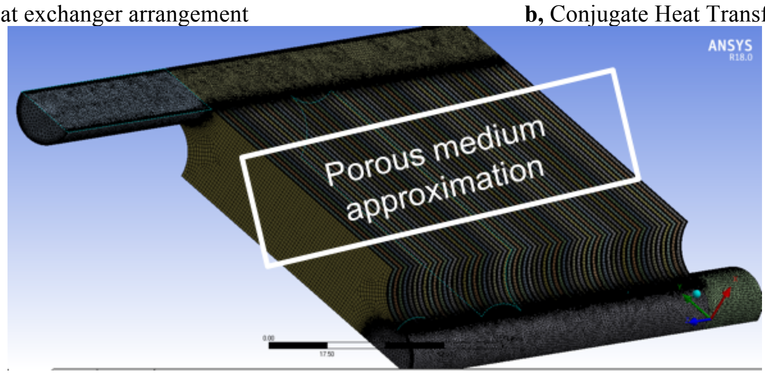
The objective of this paper is to predict micro heat exchanger performance for a micro-CHP system by a detailed modelling of complex microchannels and a new methodology to assess the entire heat exchanger characteristics based on reduced order modelling. The heat exchanger is made from a pile of counterflow flow passages with optimised thickness separated by thin foils. The flow passages are delimited by frames with integrated collectors. Geometries of both frames are identical, and they are positioned on top of each other by mirroring. A metallic wire mesh is inserted in the flow passages to provide the required thickness and stiffness for the microchannels. A metallic wire mesh is inserted in the flow passages to provide the required sti↵ness and microchannel efficiency enhancement. Due to the complexity of the geometry and intricate channels, it is difficult to numerically simulate the entire heat exchanger. Appropriate CFD hypotheses are required to reduce the very high computational cost which would result from a full heat exchanger model. The new CFD methodology (comprises of Conjugate Heat Transfer models and Reduced order models) could reduce the computational size to a considerable large extend (billion cells to few million cells) with good accuracy. Each micro channel mesh size of the reduced model is very coarse (approx. 10000 cells) as compared to CHT model mesh (approx. $10*10^7$ cells), to mimic the microchannel characteristics. Counterflow microchannel arrangement (Figure 1, a), computational Conjugate Heat Transfer domain (Figure 1, b) together with reduced model (collectors with microchannels, Figure 1, c), is shown in Figure 1.
The reduced model consists of collectors and microchannels, modelled through a porous medium approximation [1]. The porous medium model, based on Darcy-Forchheimer law [2,3] is modified (Constant Integration Method) to account for the temperature evolution in the heat exchanger. The resulting microchannel

characteristics (Pressure drop, temperature drop etc.) from a series of 3D CFD-Conjugate Heat Transfer analysis is used to calculate the inertial and viscous coefficients [4] using the Constant integration method. The reduced model was characterized for both Inverted Brayton Cycle (IBC) and Brayton Cycle (BC) operating conditions.

As the thermal efficiency varies based on inlet mass flow, consequently inertial and viscous coefficients also differ based on inlet mass flow. Least Square fit and Extremum method were introduced to fit an average from the inertial and viscous coefficients curve for all the 60 microchannels. User-defined function method is implemented to provide a varying inertial and viscous coefficients based on inlet mass flow for all the 60 microchannels. All three methods have been implemented and verified for both IBC and BC operating conditions. The best-revised methodology allows obtaining pressure losses with less than three percent error with respect to the 3D CFD-Conjugate Heat Transfer modelling (See Figure 2, a). Mass flow deviation from the corresponding CHT inlet mass flow for the 60 microchannels (with collectors, see Figure 1,c) calculated using the reduced order modelling is depicted in Figure 2, b.

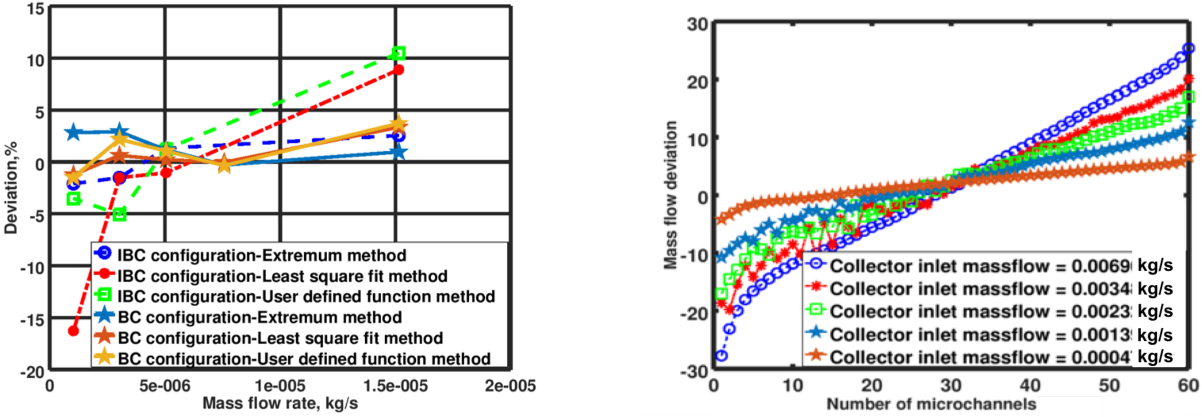


a, Counter flow heat exchanger arrangement **b,** Conjugate Heat Transfer computational



c, Computational domain for collectors with micro channels.

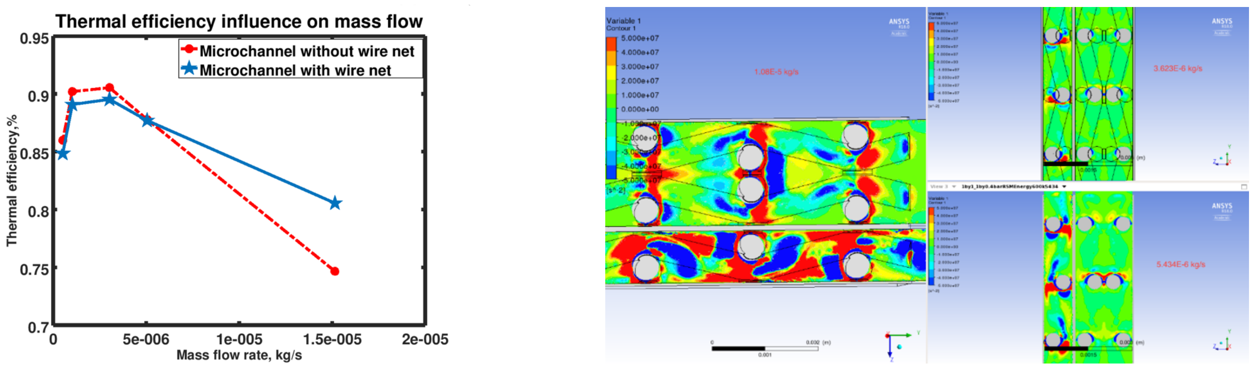
Figure 1



a, Reduced model validation for different operating conditions **b,** Mass flow distribution

Figure 2

A parametric study was carried out considering the effect of the wire mesh geometry, foil thickness and microchannel length. The Reynolds number based on inlet is between 30-300 (for micro-channels), 2000-30000 (for secondary collectors) and even higher for primary collectors. Microchannel porosity (75%) can enhance localized turbulence at smaller Reynolds numbers. Since the Knudsen number is low, thermal creep or rarefaction effects are negligible. An optimum microchannel mass flow is found to ensure the highest thermal efficiency (see Figure 3, a) and seems to be quite independent of the microchannel parameters. It is a function of localized turbulence [5] produced near the net intersections. Figure 3, a shows a heat transfer enhancement (relative to the microchannel without wire net) at higher mass flows. This is due to the strong counter rotating vortices at higher mass flows which ensures localized mixing and thereby enhances heat transfer (see Figure 3, b). Higher order turbulence model (Reynolds stress model) was used to investigate the effect of high wall normal fluctuations (turbulence production terms) on heat transfer enhancement at various mass flows.



a, Optimum mass flow and efficiency increase due to wire net **b**, Lambda 2 vorticity shows enhanced mixing at higher mass flows

Figure 3

Acknowledgements

This project has received funding from the European Union's Horizon 2020 research and innovation programme under the Marie Skłodowska-Curie grant agreement No. 643095.

References and Citations

[1] Yizhou Y., Rizwanzuddin N. S. (2005). *CFD simulation of a research reactor*, Mathematics and Computation, Supercomputing, Reactor Physics and Nuclear and Biological Applications, September 12-15, 2005.
[2] D.A. Nield, A. Bejan, (2013). *Convection in Porous Media*, 4th edition (Springer Inc., New York, 2013).
[3] Girault, V. & Wheeler, M.F. Numer. Math. (2008). *Numerical discretization of a Darcy–Forchheimer model.* 110: 161. doi:10.1007/s00211-008-0157-7.
[4] Nicolas G. (2011). *High temperature and pressure reactive flows through porous media. International Journal of Multiphase Flow*, Elsevier, 2011, 37 (1), pp.24-35.
[5] Peter S. Bernard & James M. Wallace (2002). *Turbulent Flow: Analysis, Measurement & Prediction*. John Wiley & Sons. p. 324. ISBN 0471332194.1.