Modeling an Open-Cell Foam Heat Exchanger

B. Chinè[1], F. Rodriguez-Mendez[1], M. Meneses-Guzman[2]

[1] School of Materials Science and Engineering, Costa Rica Institute of Technology, Cartago, Costa Rica
[2] School of Industrial Production Engineering, Costa Rica Institute of Technology, Cartago, Costa Rica

bchine@itcr.ac.cr

October 22 - 24

Presentation overview

- Open-cell metal foam, heat exchanger
- Geometry and governing equations
- Numerical results
- Conclusions

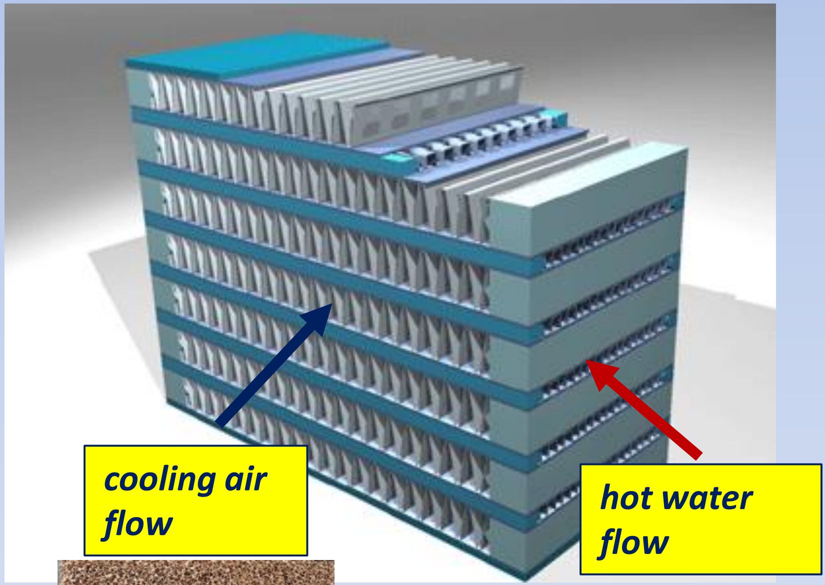
Open-cell metal foam, compact heat exchanger

- Open-cell metal foams (or metal sponge) can be used to enhance heat transfer in many applications, such as cryogenic heat exchanger, **compact heat sinks and heat exchanger.**

- They are characterized by a **cellular structure** represented by a metal (or a metal alloy) and connected gas voids inside.

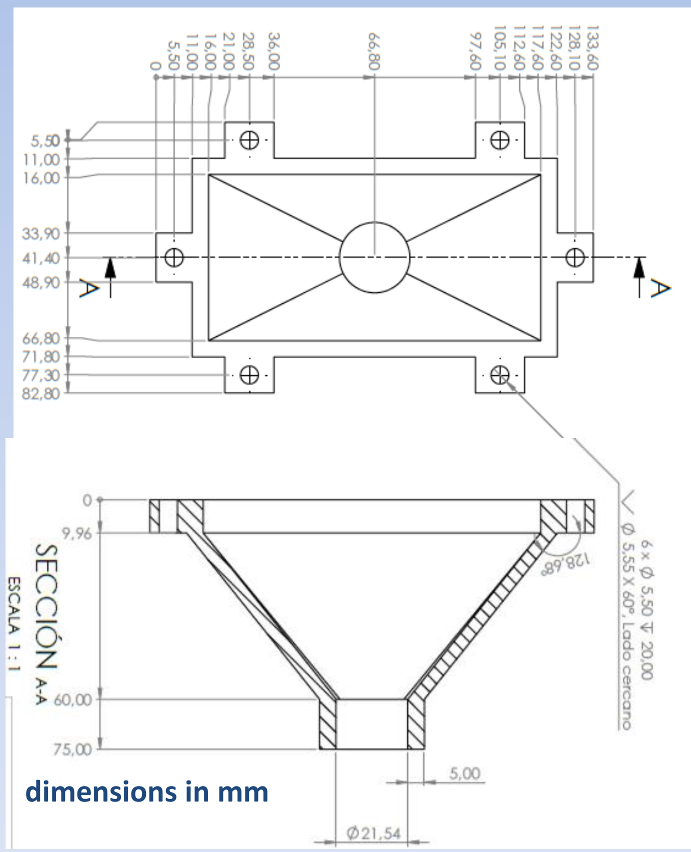
- Due to their intrinsic **high porosity and large specific surface area,** these materials are considered to have very promising properties to improve efficiency and minimize the required weight and volume of **novel industrial heat exchangers.**

- In this work, we use COMSOL Multiphysics® 6.2 to study fluid flow and heat transfer processes through a open-cell foam heat exchanger prototype.



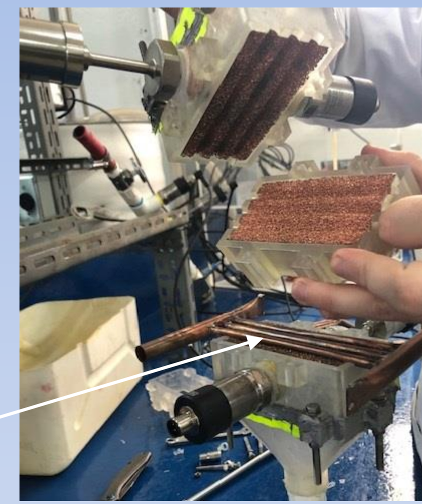
Geometry of the open-cell foam heat exchanger

end sections of the heat exchanger

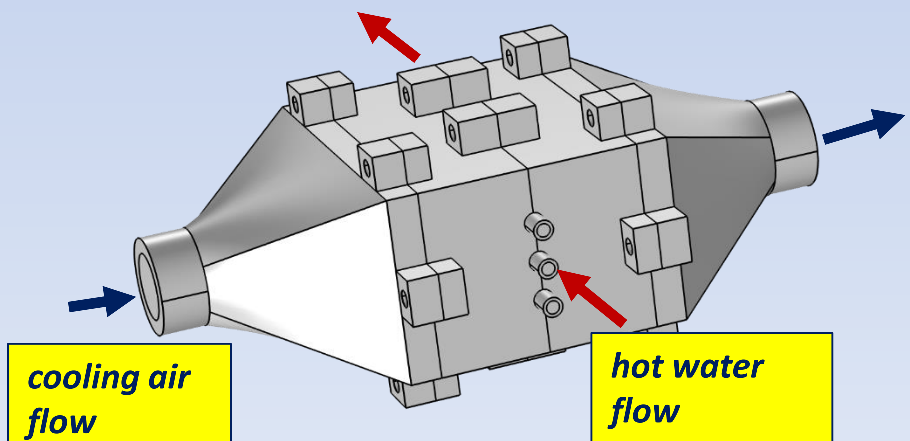


dimensions in mm

Magnitude	Value
Length L	230 mm
Width W	101.6 mm
Height H	50.8 mm
Wall thickness t	5 mm
Inner diameter D_a of air inlet and outlet	21.54 mm
Inner diameter of the copper tubes for the water crossflow	4.35 mm
Copper sponge width w	101.6 mm
Copper sponge height h	50.8 mm
Copper sponge thickness s	12.7 mm

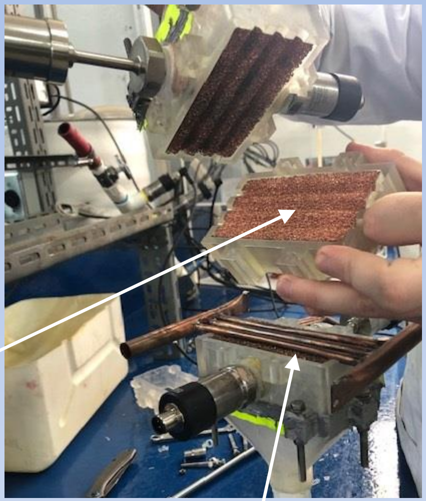


copper tubes for the water flow



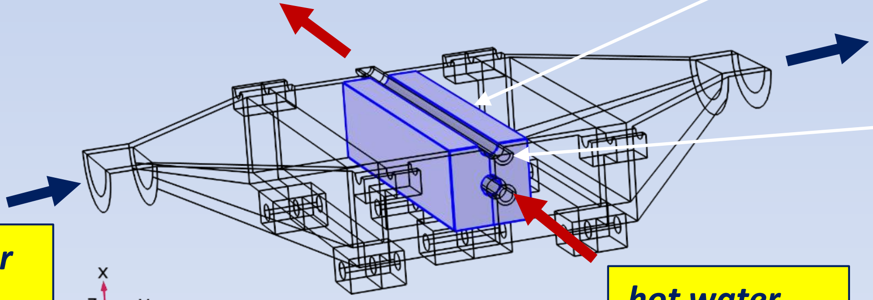
Open-cell metal foam sandwiched to three copper tubes

Magnitude	Value
Copper alloy	C10100
Pore density (pores per linear inch)	40
Relative density RD	10%
Specific heat	0.385 J/(g K)
Porosity ε_p	90%
Specific surface area S_b	24 in^2/ in^3
Total thermal conductivity	13.2 W/ (m K)
Permeability	1.6 x10^{-7} m^2
Interstitial heat transfer coefficient h_{sf}	140 W/ (m^2 K)



copper foam

copper tubes for the water flow



cooling air flow

hot water flow

compact heat exchanger reduced by symmetry

Governing equations of the model

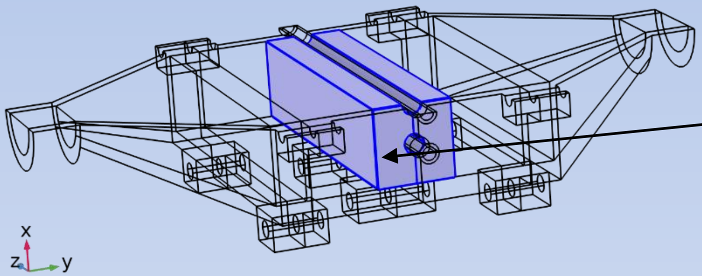
$$\nabla \cdot (\rho \mathbf{u}) = 0$$

$$\rho(\mathbf{u} \cdot \nabla)\mathbf{u} = \nabla \cdot [-p\mathbf{I} + \mu [\nabla\mathbf{u} + \{\nabla\mathbf{u}\}^{\mathrm{T}}] - \frac{2}{3}\mu[\nabla \cdot \mathbf{u}]\mathbf{I} + \mathbf{F}$$

$$\rho c_p \, \mathbf{u} \cdot \nabla \mathrm{T} + \nabla \cdot \mathbf{q} = Q$$

Heat transferred from a laminar, incompressible stream of hot water to a laminar, compressible flow of cold air.

Steady state compressible fluid flow and heat transfer through the 3D heat exchanger section (Mass and Linear Momentum Conservation; Thermal Energy Conservation).



Copper foam modelled as a porous medium with non-Darcian flow and in LTNE conditions.

$$\frac{\rho}{\varepsilon_p}(\boldsymbol{u} \bullet \nabla)\frac{\boldsymbol{u}}{\varepsilon_p} = -\nabla p + \nabla \bullet [\frac{1}{\varepsilon_p}\left\{\mu(\nabla\boldsymbol{u} + (\nabla\boldsymbol{u})^T - \frac{2\mu}{3}(\nabla \bullet \boldsymbol{u})\boldsymbol{I}\right\}] - \left[\kappa^{-1}\mu + \rho\beta|\mathbf{u}| + \frac{Q_m}{{\varepsilon_p}^2}\right]\mathbf{u} + \mathbf{F}$$

Brinkman equations

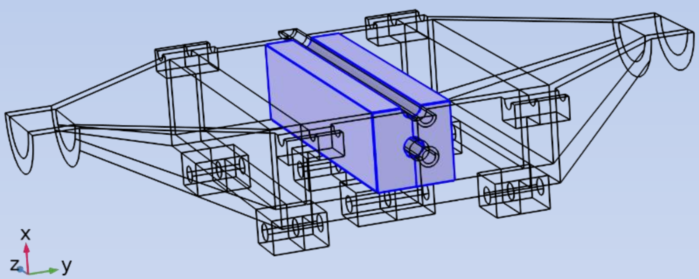
additional source term (LTNE)

$q_{sf}\left(T_s - T_f\right)$ for fluid thermal energy conservation

$q_{sf}\left(T_f - T_s\right)$ for solid thermal energy conservation

COMSOL Multiphysics® 6.2: Heat Transfer and CFD modules
Conjugate Heat Transfer physics interface

Boundary conditions



Boundary conditions for the fluid flow:

- a total flow rate of 1.893 L/min for water (inlet velocity $U_{in,w}$ = 0.708 m/s in each tube)
- a flow rate of 23.408 L/min for the cooling air (inlet velocity $U_{in,a}$= 1.071 m/s)
- at the outlets, a null gauge pressure
- boundary condition of no slip on the solid walls
- conditions of symmetry on the central longitudinal *y-z* plane of the device

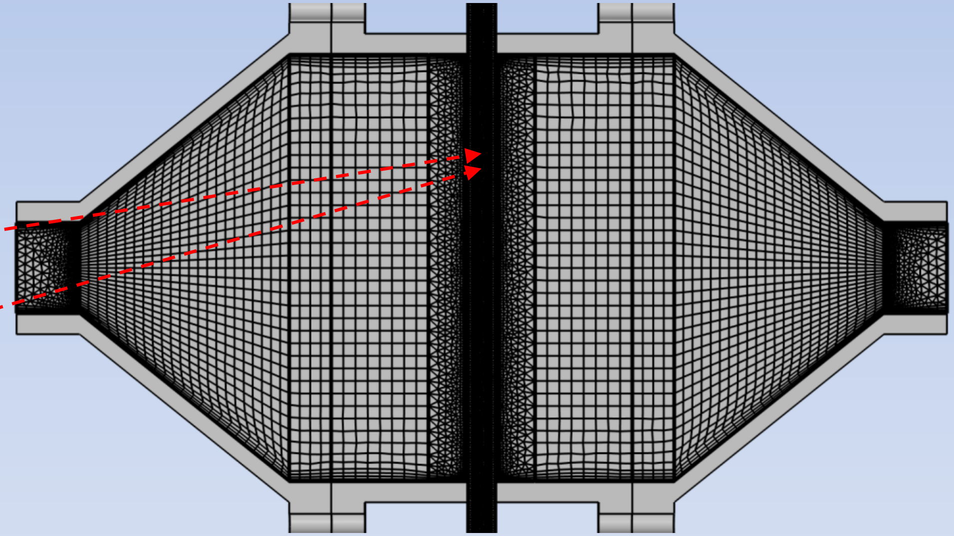
Boundary conditions for the heat transfer:

- at the inlet, temperature $T_{in,a}$ of 300 K for the air inflow and temperature $T_{in,w}$ of 312 K for the water inflow
- at the outlets $-\mathbf{n} \bullet \mathbf{q} = 0$ for both fluids (**q** is the heat flux and **n** is the normal direction)
- local thermal non-equilibrium boundary condition is used for the walls adjacent to the porous foam domain
- conditions of symmetry on the central longitudinal *y-z* plane of the device
- no thermal resistance is considered between the copper tubes and the porous medium
- conditions of thermal insulation on the rest of the surfaces

Solution with COMSOL Multiphysics 6.2

- free tetrahedral mesh in the air inlet and outlet sections, porous medium, and in the cylindrical tubes
- rest of the device is divided in structured quadrilateral elements
- boundary layers on the solid walls, using default values of the software

Parameter	Size
maximum element size	3.94 mm 2.75 mm (tubes)
minimum element size	0.744 mm 0.298 mm (tubes)

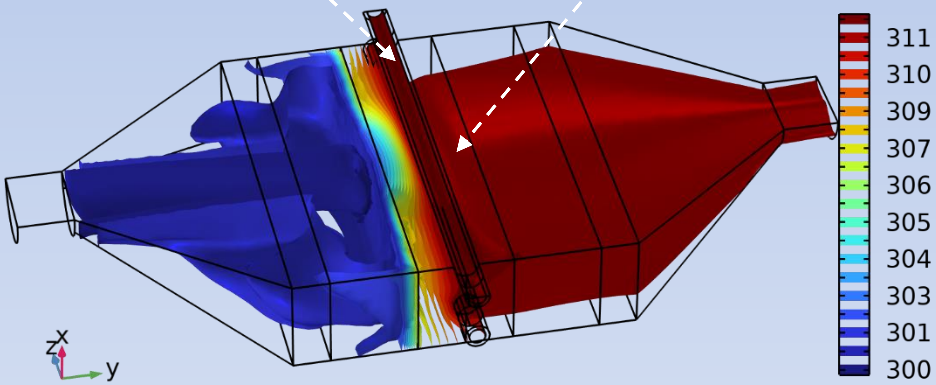


the number of degrees of freedom is approximately 6.7×10^5 plus 6×10^4 internal DOFs

Numerical results: temperature and velocity

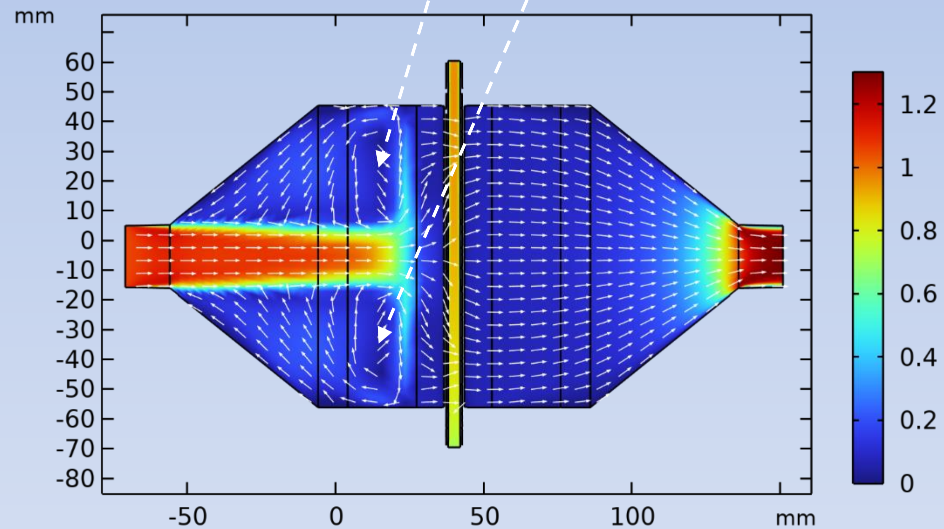
water temperature doesn't show a significant variation

air temperature increases through the open-cell foams up to a value close to 312 K



temperature iso-surfaces (K) in the heat exchanger

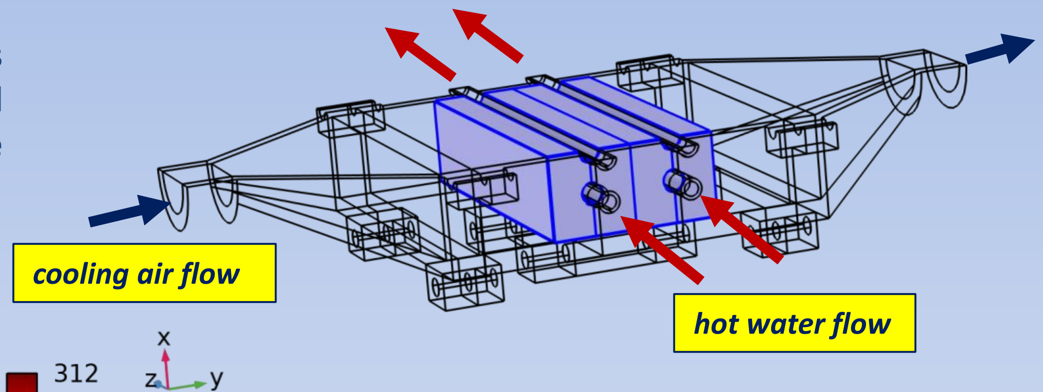
two main recirculating streams between the central air jet and the solid walls



velocity magnitude (m/s) and velocity arrows on the longitudinal central y-z plane

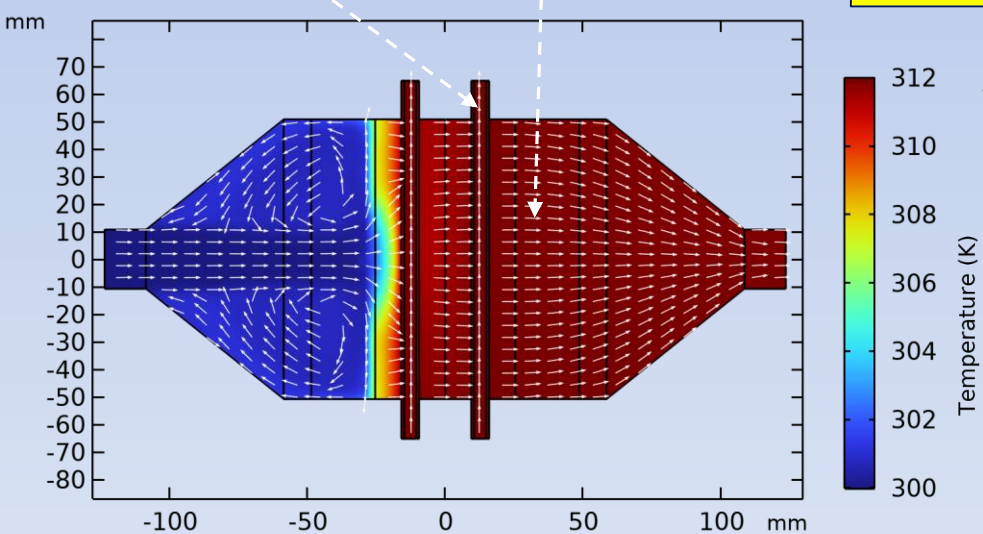
Numerical results: temperature and velocity arrows

a second series of three cylindrical tubes for the water flow added, but maintaining the same flow rate

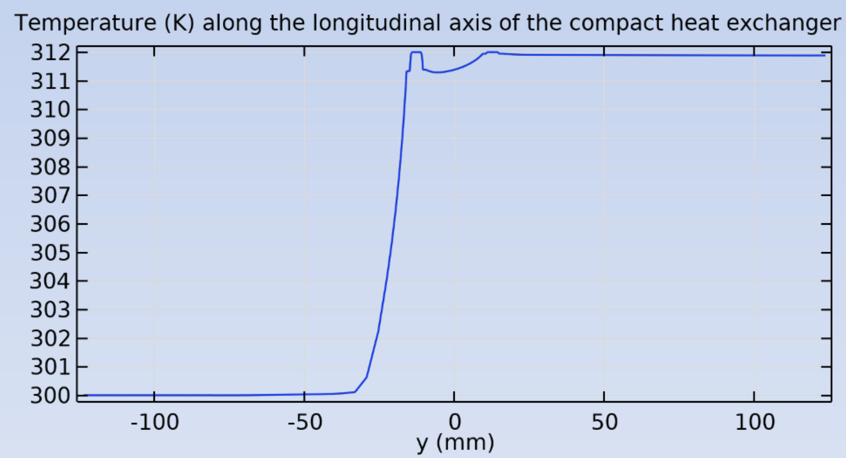


again, water temperature doesn't show a significant variation

air temperature increases through the open-cell foams up to a value close to 312 K



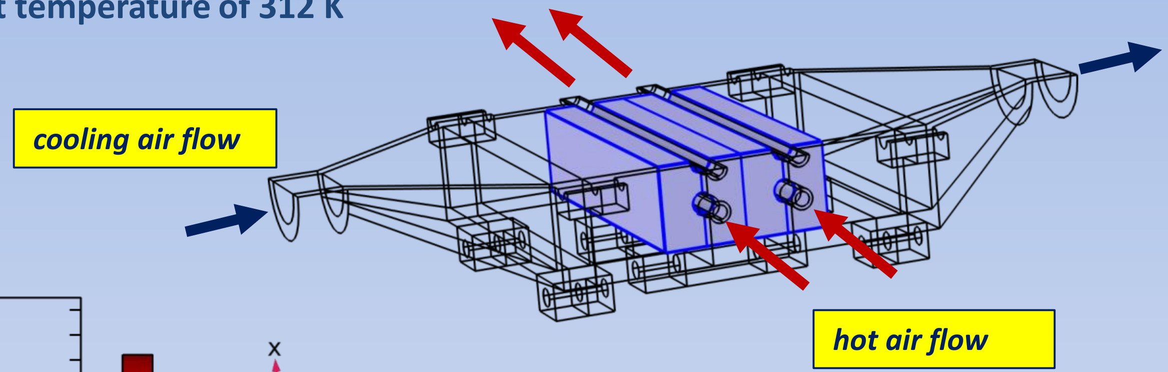
temperature (K) and velocity arrows on the longitudinal central y-z plane (heat exchanger with 6 tubes for the water flow)

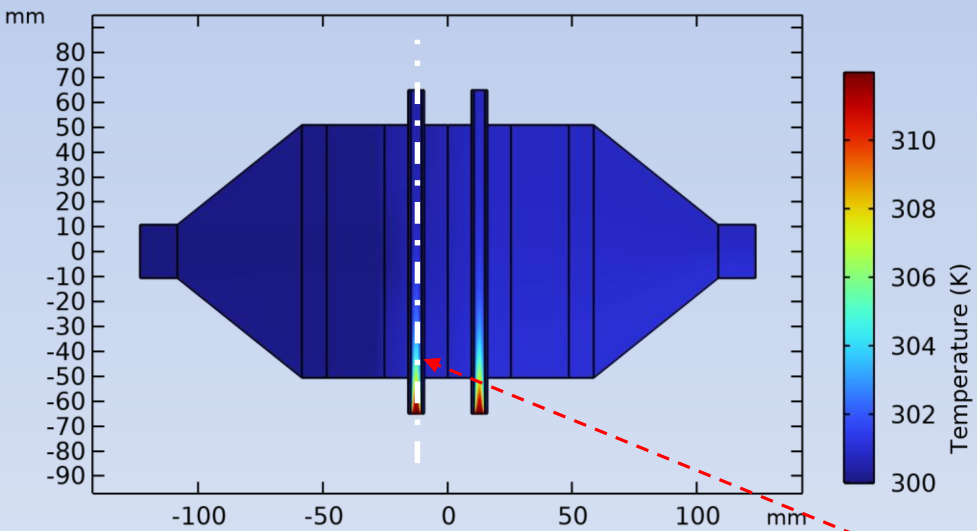


temperature (K) along the longitudinal y axis of the heat exchanger, with 6 tubes for the water flow

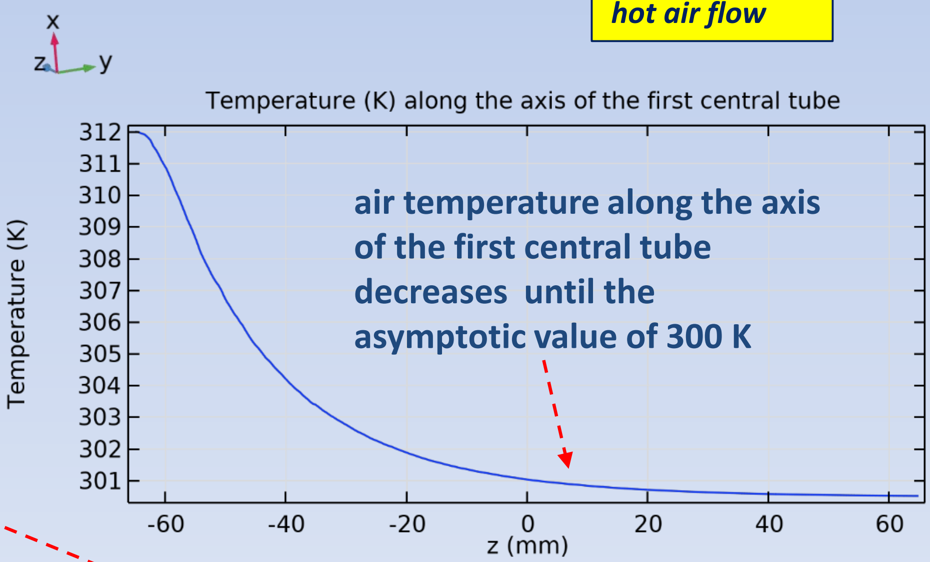
Numerical results of temperature

replacing water with air at the same inlet temperature of 312 K





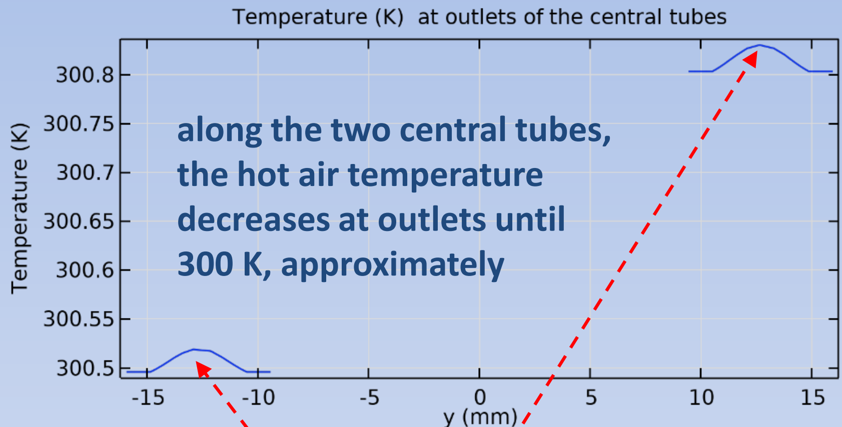
temperature (K) on the longitudinal central z-y plane (heat exchanger with 6 tubes for air-air heat transfer)



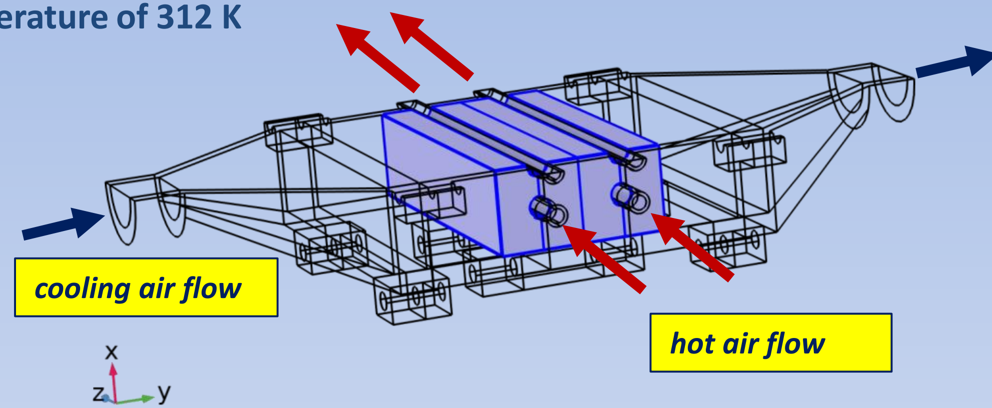
temperature (K) along the z axis of the first central tube (heat exchanger with 6 tubes for air-air heat transfer)

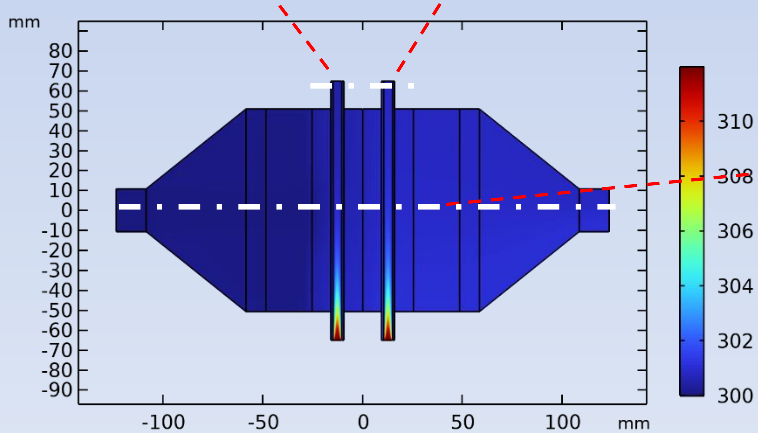
Numerical results of temperature

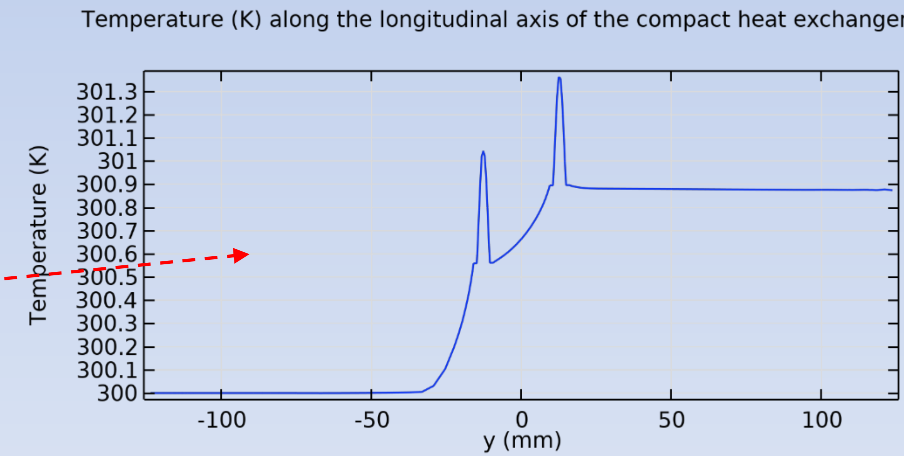
replacing water with air at the same inlet temperature of 312 K



along the two central tubes, the hot air temperature decreases at outlets until 300 K, approximately



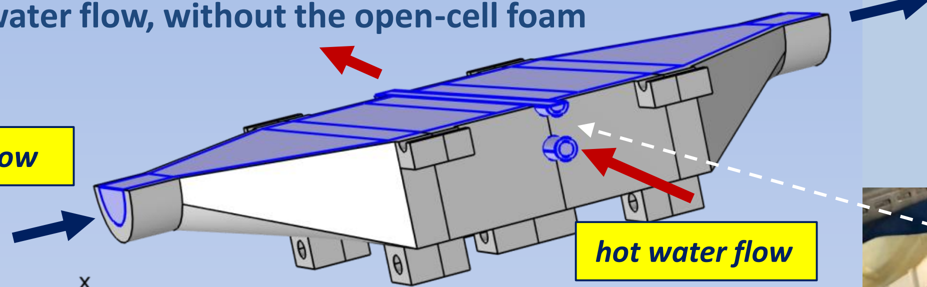


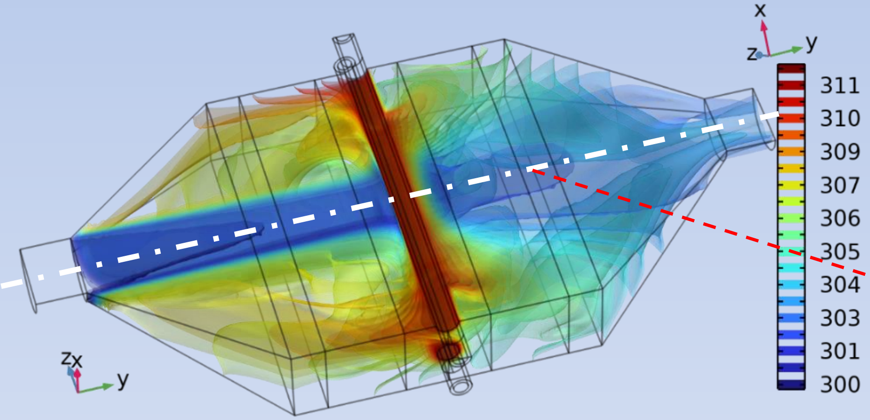


cold air temperature increase of 1 ℃ approximately, a much lesser value than 11 ℃ , as computed before for the heat exchanger with water as hot fluid

Numerical results of temperature

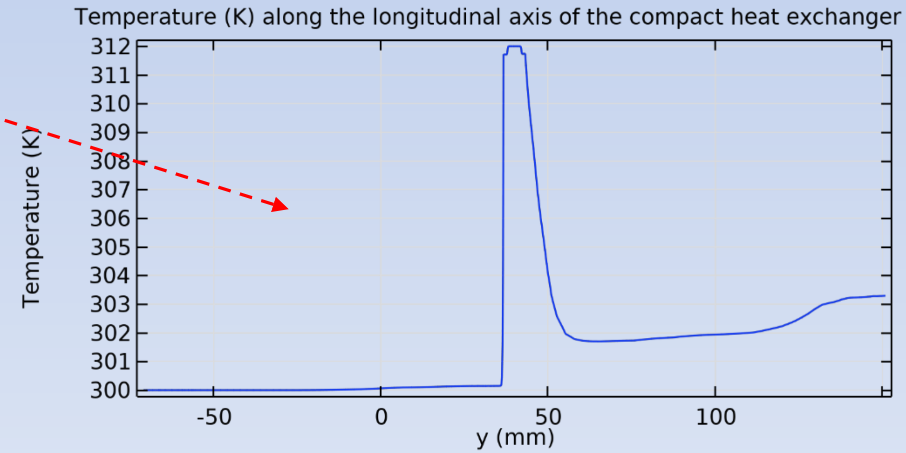
with three nude copper tubes for the water flow, without the open-cell foam





air temperature difference between the inlet and the outlet is approximately 3 °C

nude copper tubes for the water flow, without foam



According to the computational results, the energy transfer of the exchanger was enhanced using a copper sponge (high porosity and large specific surface area)

Conclusions

- It has been developed a computational model of the fluid flow and heat transfer in a 3D prototype of an open-cell foam heat exchanger.

- The numerical findings of the simulations show that the energy transfer of the exchanger is enhanced owing to the copper sponge's high porosity and large specific surface area.

- Despite of the improved heat transfer experimented by the cooling air, the reduction of the hot water temperature is low, also constrained by the short tube length, therefore requiring some improvements of the device (introduction of coils, narrow channels, etc.).

- The results, also confirmed by experimental work, show that the computational model developed with COMSOL Multiphysics® is effective for modelling the conjugate heat transfer process of this compact device.

References

[1] F. Garcia-Moreno, "Commercial Applications of Metal Foams: Their Properties and Production", *Materials*, 9, 85; doi:10.3390/ma9020085, 2016.
[2] T.J. Lu, H.A. Stone and M.F. Ashby, "Heat transfer in open-cell metal foams", *Acta Materialia*, 46(10), 3619-3635, 1998.
[3] K. Boomsma, D. Poulikakos and F. Zwick, "Metal foams as compact high performance heat exchangers", *Mechanics of Materials,* 35, 1161-1176, 2003.
[4] A. Kopanidis, A. Theodorakakos, E. Gavaises and D. Bouris, "3D Numerical simulation of flow and conjugate heat transfer through a pore scale model of high porosity open cell foam", *Int. Journ. of Heat and Mass Transfer,* 53, 2539-2550, 2010.
[5] H. Wang, L.J. Guo and K. Chen, "Theoretical and experimental advances on heat transfer and flow characteristics of metal foams", *Sci. China Tech. Sci*, 63, 705-718, 2020.
[6] S. Meinicke, T. Wetzel and B. Dietrich, "CFD Modeling of single-phase hydrodynamics and heat transfer in solid sponges", *11th Int. Confer. on CFD in the Minerals and Process Industry-CSIRO*, Melbourne, Australia, 7-9 December 2015.
[7] T. Dixit and I. Ghosh, "Simulation intricacies of open-cell metal foam fin subjected to convective flow", *App. Thermal Engin.*,137, 532-544, 2018
[8] A. Jafarizade, M. Panjepour, M. Meratian and M.D. Emami, "Numerical simulation of gas/solid heat transfer in metallic foams: A general correlation for different porosities and pore sizes", *Transport in porous media,* 122(3), DOI 10.1007/s11242-018-1208-x, 2018.
[9] A.M Bayomy. and M.Z. Saghir, "Experimental and numerical study of aluminium metal foam (with/without channels) subjected to steady water flow", *Pertanika Journal of Science and Technology*, **25**(1), 221-246, 2017.
[10] B. Bidar, F. Shahraki and D. M. Kalhori, "3D Numerical modelling of convective heat transfer through two-sided vertical channel symmetrically filled with metal foams", *Periodica Polytechnica Mech. Engineer.*, 60(4), 193-2012, 2016.
[11] M. Odabaee and K. Hooman, "Metal foam heat exchangers for heat transfer augmentation from a tube bank", *App. Thermal Engin.*, 36, 456-463, 2012.
[12] S. Mancin, C. Zilio, A. Cavallini and L. Rossetto, "Heat transfer during air flow in aluminum foams", *Int. Journ. of Heat and Mass Transfer,* 53, 4976-4984, 2010.
[13] A.M. Hassan, A.A. Alwan and H.K. Hamzah, "Numerical study of fan coil heat exchanger with copper-foam", *Int. Journ. of Fluid Machinery and Systems*, 16 (1), January-March, 2023.
[14] K. Yang, K. Liu and J. Wang, "Pore-scale numerical simulation of convection heat transfer in high porosity open-cell metal foam under rotating conditions", *Applied Thermal Engineering*, 195, 117168, 2021.
[15] B. Pulvirenti, M. Celli and A. Barletta, "Flow and convection in metal foams: A survey and new CFD results", Fluids, 5, 155; doi:10.3390/ fluids5030155, 2020.
[16] K. Kumar, B. Kotresha and K. Naik, "Flow and heat transfer irreversibility in partial filled metal foams", *Int. Journ. of Thermal Sciences,* 184, 107968, 2023.
[17] N. H. Roge, K. Yogi, A. Singh, H. Shrigondekar, S. Krishnan and S.V. Prabhu, "Local heat transfer distribution of thermally developing region in a rectangular open-cell metal foamed channel", *Int. Journ. of Thermal Sciences,* 203, 109138, 2024.
[18] Comsol AB, Comsol Multiphysics-CFD Module, *User's Guide*, Version 6.2, 2023.
[19] Comsol AB, Comsol Multiphysics-Heat Transfer Module, *User's Guide*, Version 6.2, 2023.
[20] T.L. Bergman, A.S. Lavine, F.P. Incropera and D.P. Dewitt, *Fundamentals of heat and mass transfer,* 7 ed., John Wiley and Sons, Hoboken (NJ), USA, 2011.

Acknowledgements

Many thanks for your attention !

We would like to also acknowledge:

TEC Tecnológico de Costa Rica **Vicerrectoría de Investigación y Extensión**

COMSOL CONFERENCE 2024 FLORENCE