COMSOL CONFERENCE 2019 BOSTON

# Experimental and Numerical Evaluation of Metal Foam Thermal Performance Interacting with Nanofluid: Applications in Electronics Cooling

C. A. Welsford, C. S. Delisle, M. Z. Saghir
Mechanical Engineering, Ryerson University, Toronto, ON, Canada

## Introduction

The use of both nanofluids and foam metals in the cooling of thermal systems is a topic of rapidly growing interest. The high surface area to volume ratio offered by both system alterations shows great potential to enhance the rate of removal of thermal energy. Further, there exists nanoscale phenomena which have been observed to further enhance the strength of the heat transfer mechanism present within a particular Nanofluid. Within the foam metals, the highly tortuous nature of the material creates a tendency for mixture to occur. This assists in the absorption of thermal energy into the working fluid. However, with both additions there exist phenomena on the sub-macro-scale which greatly influence the thermal systems behavior. These phenomena have posed as large barriers for the widescale application of both technologies via difficulties in modeling and simulation. The objective of the following is to compare the numerical results obtained using COMSOL Multiphysics and the Nanofluid properties obtained by Ho et. al. [1] with empirically obtained results. This comparison will offer insights into the use commercial CFD software for the simulation of both nanofluids and foam metals.

## Theory / Experimental Set-up

The modeling and simulation of any multi-phase fluid flow is complicated. However, it becomes more complicated when considering a phase operating at the nanoscale. The nanoparticles suspended within the mixture presently started are to the order of 50 nm or less. When the suspended particles reach this scale certain phenomena such as thermophoresis, Brownian motion, and many more come into play and greatly affect the behavior of the fluid as a whole. A popular approach to circumvent the modeling of these complex fluid mechanisms is the modeling of the fluid as a single-phase fluid with analogous properties. The work done by Ho et. al. [1] evaluated the properties of the nanofluids experimentally and developed a correlation of the properties with respect to the fluid Nanoparticle concentration for alumina nanoparticles suspended in water. The present work uses these established properties combined with experimental work based on the system designed by Bayomy et. al. [2] to evaluate the effectiveness of COMSOL Multiphysics and the analogous properties from Ho et. al. to model nanofluids within foam metals.

The model considered at present is formed of three porously filled channels. The metal foam used to fill the channels has a porosity of 0.91 and a linear pore density of 40 pores per inch. The foam samples are guaranteed by the manufacturer to be continuous and homogeneous in nature.

## Governing Equations / Numerical Model / Simulation

The present paper considers the three-dimensional laminar single-phase flow of nanofluids through highly porous open-cell foam metals. The model is considered to be operating under local thermal equilibrium conditions, and to be steady state.

The model employs two integrated modules from COMSOL Multiphysics. The first is the free and porous flow module which was used to model the laminar flow of the fluid through the test section. This includes the general flow area and the area composed of bulk porous media. The second module employed was the heat transfer in porous media module. This module allowed simulation of the movement of energy through the test section. The module was used to cover solid, liquid, and fluid-saturated porous media regimes. Figure 1 shows the domains employed in the system. Domains 1, 3, and 5 correspond to free fluid flow governed by the general Navier-Stokes equation shown in Equation 1

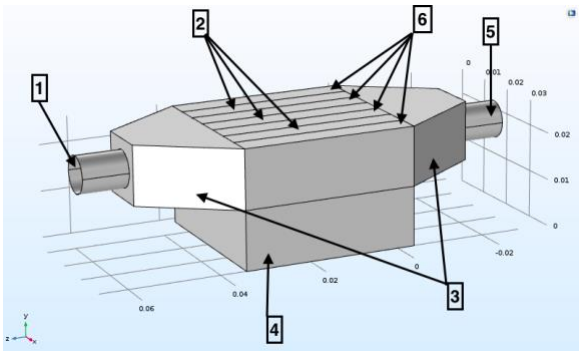


**Figure 1.** Computational domains employed in the present study.

**Equation 1.** $\rho(\mathbf{u}\cdot\nabla)\mathbf{u} = \nabla\cdot[-p\boldsymbol{I} + \mu(\nabla\boldsymbol{u} + (\nabla\boldsymbol{u})^{\boldsymbol{T}})] + \boldsymbol{F}$

Domains 4 and 6 denote the solid phases of the system. The interaction between the fluid and these solid phases is considered to be non-slip. Domain 2 denotes the fluid-saturated porous medium. The fluid flow within this regime is governed by the Brinkman-Forchheimer equation as shown in Equation 2.

**Equation 2.** $\frac{\rho}{\epsilon_p}\left((\boldsymbol{u}\cdot\nabla)\frac{\boldsymbol{u}}{\epsilon_p}\right) = \nabla\cdot[-p\boldsymbol{I} + \frac{\mu}{\epsilon_p}(\nabla\boldsymbol{u} + (\nabla\boldsymbol{u})^{T}) - \left(\mu k^{-1} + \beta_F|\boldsymbol{u}| + \frac{Q_m}{\epsilon_p^2}\right)\boldsymbol{u} + \boldsymbol{F}$

The heat transfer within the system is governed by the energy equation with the addition of an effective thermal conductivity within the porous domain. This is represented by equation 3.

**Equation 3.** $\rho c_p \boldsymbol{u}\cdot\nabla T_2 + \nabla\cdot\boldsymbol{q} = Q + Q_p + Q_{vd}, where\ \boldsymbol{q} = -K_{eff}\nabla T_2$ and $k_{eff} = \theta_p k_p + (1-\theta_p)k + k_{disp}$

The boundary conditions of the system were defined as follows. The inlet was set to be isothermal with a constant fluid velocity. The bottom of the large solid surface, denoted by 4, was to set to be a general inward heat flux. The outlet was set to be free and open for both mass and energy. This was done to conserve system continuity. All other boundaries were set to be insulated for energy transfer and non-slip for all walls. The system conditions above were set to match the measured empirical values of the experiment. The contact resistance between the foam and the heater was modeled using a thin layer with a fixed conductivity.

The Nanofluid properties obtained by Ho et. al. [1] which were used in the present study, for a concentration of 0.3% by volume, are as follows: viscosity of 0.001019 kg/ms, density of 1006.005 Kg/m$^3$, specific heat of 4145.316698 J/kgK, and a thermal conductivity of 0.618523 W/mK [1].

## Experimental Results / Simulation Results / Discussion

As mentioned previously the objective of the present was to determine the effectiveness of COMSOL Multiphysics combined with the nanofluid properties obtained by Ho et. al. [1] to accurately model flow through foam metal filled channels.

The experimental and numerical temperature distributions are shown in Figure 2 below for a constant flow rate of 0.2 USGPM. It can be seen clearly that there exists good agreement between the experimentally and numerically obtained results, with a maximum relative error of 4.3%.

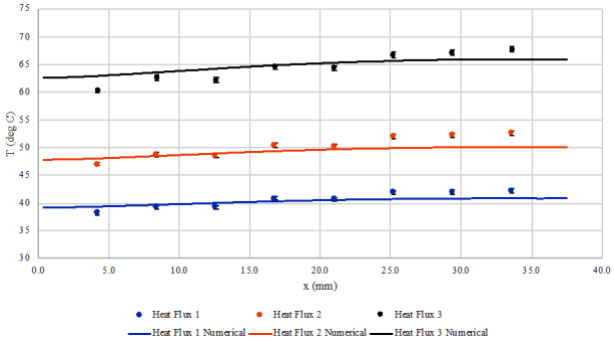


**Figure 2.** Temperature distribution using nominal heat fluxes of 50000W/m$^2$ (1), 75000 W/m$^2$ (2), and 100000W/m$^2$

The curve shows a clear increase in the temperature distribution of the system as the heat flux increases. This observation helps to reassert the accuracy of the simulation from a physical perspective.

Further, COMSOL offers insights into the fluid behavior within the system. Measurements of fluid velocity and pressure across open-cellular metal foams are extremely difficult without influencing the behavior of the fluid, as such COMSOL offers a meaningful alternative. The velocity contours and magnitudes are shown in Figure 3 below.

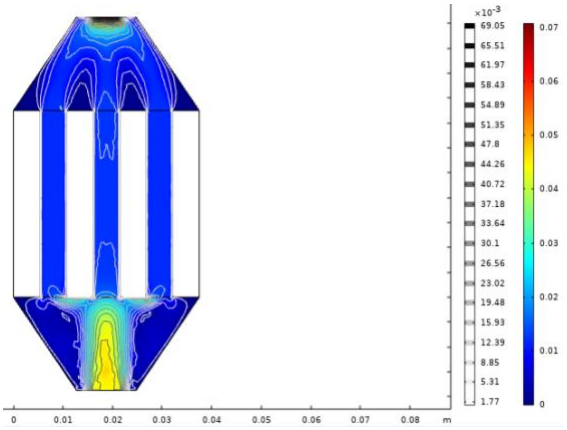

**Figure 3**. Numerically obtained fluid velocity through simulated test sample.

The simulation shows a clear stacking of constant velocity contours towards the walls of the sample. This is consistent with both physical expectations and the imposed non-slip boundary condition. The observed fluid behavior can then be said to be a powerful tool when making engineering decisions regarding the sample's geometry. The addition or removal of channels can affect the stacking of the velocity contours and as such directly influence the pressure drop across the system.

## Conclusions

The above work compared the results obtained numerically from COMSOL Multiphysics to empirical values. From this several major conclusions can be drawn regarding the simulation of nanofluids and highly porous metal foam:

- COMSOL Multiphysics combined with the nanofluid properties obtained by Ho et. al. [1] was able to accurately represent an experimental system with a maximum relative error of 4.3%.
- Based on this obtained accuracy COMSOL Multiphysics can then be used to gain insight into the behavior of the fluid within a particular sample. This would otherwise not be possible using conventional methods without causing flow disruption.
- To continue developing the present work it is important to determine the effects which temperature dependency has on the effective properties of the fluid
- Further, study into the effects of local thermal non-equilibrium should be conducted to determine the effects on system modelling.

## References


[1] C. Ho, M. Chen and Z. Li, "Numerical Simulation of Natural Convection of Nanofluid In a Square Enclosure: Effects Due to Uncertainties of Viscosity and Thermal Conductivity," *International Journal of Heat and Mass Transfer*, vol. 51, no. 17, pp. 4506-4516, 2008.

[2] A. Bayomy and M. Saghir, "Experimental and Numerical Study of the Heat Transfer Characteristics of Aluminum Metal Foam (With/Without Channels) Subjected to Steady Water Flow," *Journal of Science and Technology*, vol. 25, no. 1, pp. 221-246, 2017.


## Acknowledgements


The authors would like to acknowledge the support of Ryerson University and the National Science and Engineering Research Council of Canada for their support which made the present study possible