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 objective of the present work is to assess the ability of COMSOL Multiphysics working with the Nanofluid properties established by Ho et. al. [1] to accurately model experimental Nanofluid and foam metal. The system in question operates under forced convection conditions. The system considers a general inward heat flux through a heater designed to replicate system conditions observed in modern computer processors.

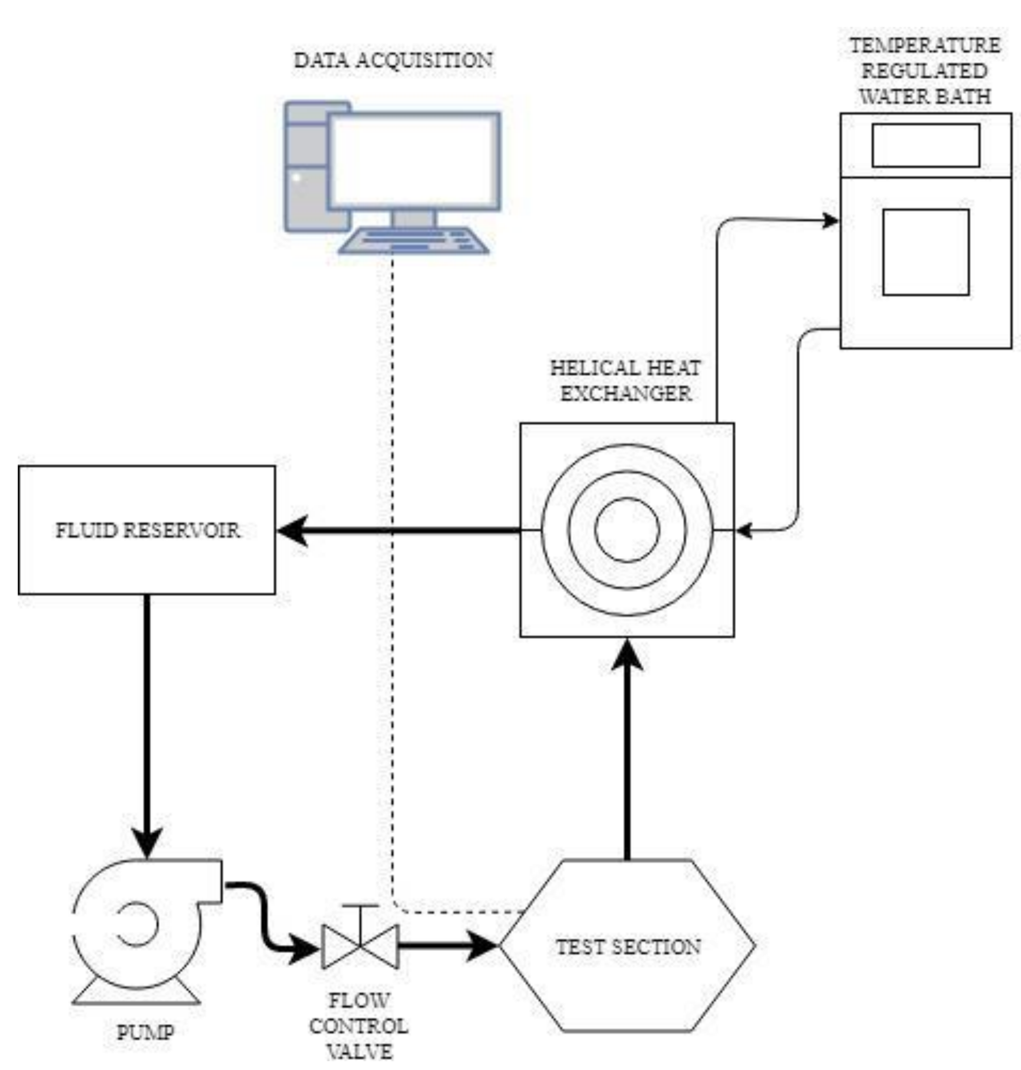

Figure 1. Schematic of Experimental Apparatus

COMPUTATIONAL METHODS: The present study employs version 5.3a of COMSOL Multiphysics to model the behavior of the system numerically. The Multiphysics modules used in the present are heat transfer in porous media and free and porous media flow. As such the study was restricted to only laminar flow rates. Figure 1 below shows the domains and boundary conditions of the model. The domains denoted by 1, 3, and 5 correspond to free fluid flow. Domains 4 and 6 compose the solid phase. Lastly, domain 2 is composed of fluid saturated porous media. The inlet boundary condition at 1 is isothermal and at a constant velocity. The outlet is free and open for both energy and mass. At the bottom of domain 4 a general inward heat flux is applied. All walls are considered to be non-slip, and all other thermal boundaries are considered insulated.

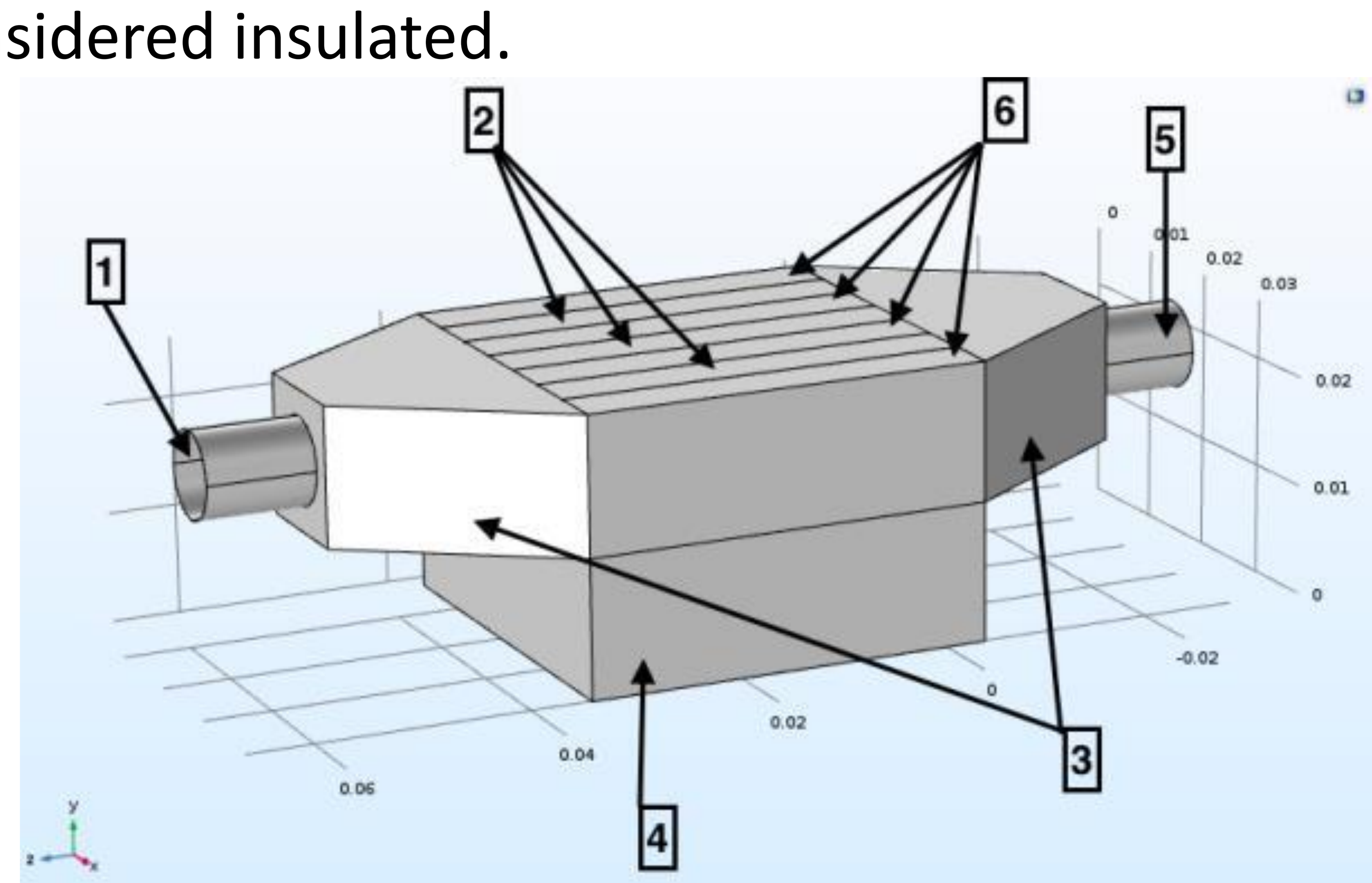

Figure 2. Numerical Domains and Boundary Conditions

RESULTS: To determine the validity of the Multiphysics simulation the numerical and experimental results were compared. The numerical conditions were set to match the experimental conditions as closely as possible based on observed properties and measured values. The properties of the Nanofluid which were used in the simulation were based on the work previously completed by Ho et. al. [1] Good agreement between the experimental and numerical results was shown with a maximum relative error of 4.3% These results are shown in Figure 3. The figure represents multiple flow rates with a constant nominal heat flux of 50000 W/m^2. The Nanofluid concentrations considered are 0.3% and 0.6% by volume. The results indicate a clear drop in the temperature distribution as the flow rate is increased. This is consistent with physical expectations. Further it can be seen that the numerical and experimental trends are in good agreement.

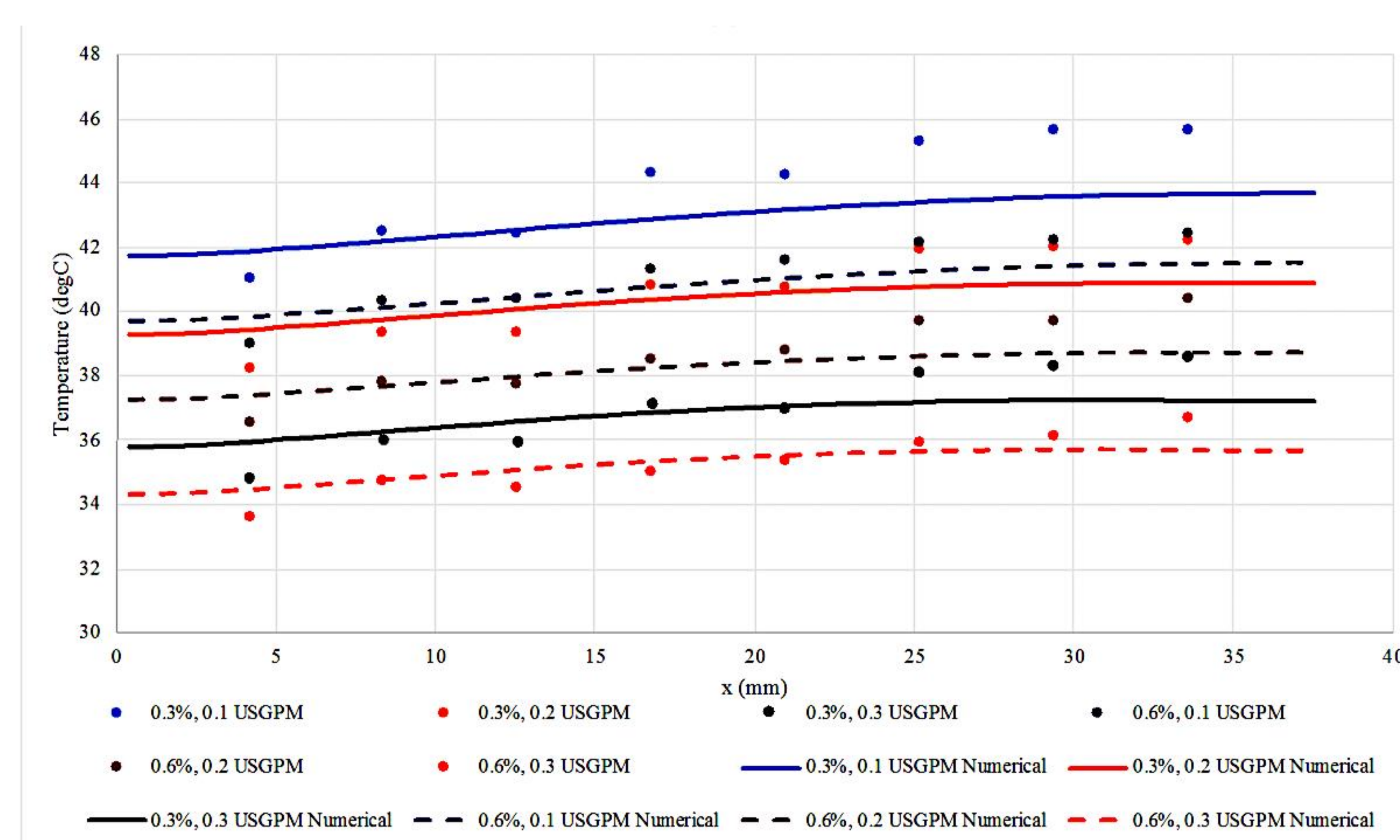

Figure 3. Schematic of Experimental Apparatus

CONCLUSIONS: The above study evaluated the ability of COMSOL Multiphysics to accurately model and experimental scenario with both Nanofluid and foam metals. The study showed good agreement between both the numerical and empirical results with a maximum error below 5%. Further the experiment showed that as the Nanofluid concentration is increased the temperature distribution of the system changes proportionally. This shows Nanofluids can prove as valuable working fluids in electronic cooling fields. To further progress the work it is advised that more study be conducted into the dependent nature of Nanofluid properties.

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.