To Design a CFD Model optimizing a Co- Designed Microfluidic Chip with Manifold Micro Channels

[1]Bablu Singh, [1]Raghvendra Gupta, [2]Sanjay Ramachandra

[1]Indian Institute of Technology Guwahati, India

[2]Indian Institute of Technology Bombay, India

ABSTRACT

The objective of this study is to investigate the microchannel cooling for microelectronic in microfluid channel by passing coolant liquid. The system operates on the principle of conjugate heat transfer, simultaneously solving the fluid flow and heat conduction in both liquid and solid regions. Numerical simulations were employed to optimize the module’s geometric design. The study primarily focuses on the influence of channel boundaries and liquid- solid interactions on flow patterns. The study investigates the effects of varying flow conditions 1 mm/s and 10 mm/s, represented by the inlet velocity. The fluid properties and geometric dimensions are such that these flow conditions result in a laminar flow regime. The results for these two cases are analysed by examining the velocity and temperature contours, as well as the pressure field, to understand the fundamental physics of conjugate heat transfer and the impact of the flow rate on the system's thermal and fluid dynamic behaviour. The fluid flow was modelled using the continuity and Navier-Stokes equations for mass and momentum conservation, while the heat transfer in liquid and solid phases was governed by separate energy equations. The governing equations were solved using the Finite Volume Method (FVM) along with formulation to handle any moving boundary or solid-liquid interaction. The PIMPLE scheme was applied for pressure-velocity coupling, GAMG was used as the pressure solver, and PBiCGStab was for solution.