Modeling of heat transfer

We have performed numerical simulation of microfludic channel using finite element analysis method (ANSYS software version11.0). Dynamic flow analysis of microfluidics channel is complex task. We have simplified the simulation by few assumptions:

1. water itself was considered to be solid material thermal properties identical to water.
2. Instead of dynamic analysis in a tube, we have replaced the space domain system with time domain. In reality that means that instead of running complex dynamic 3D model of a tube and "look" inside the tube, we have ran a transient analysis of a tube segment converting 3D problem into much simpler 2D.

The segment of the tube was meshed with shell elements. We have performed transient analysis. As boundary condition temperature of all nodes was set to 25^{o}C. At time t=0s temperature of the tube was changed to 100^{o}C and temperature in the tube centre was observed.

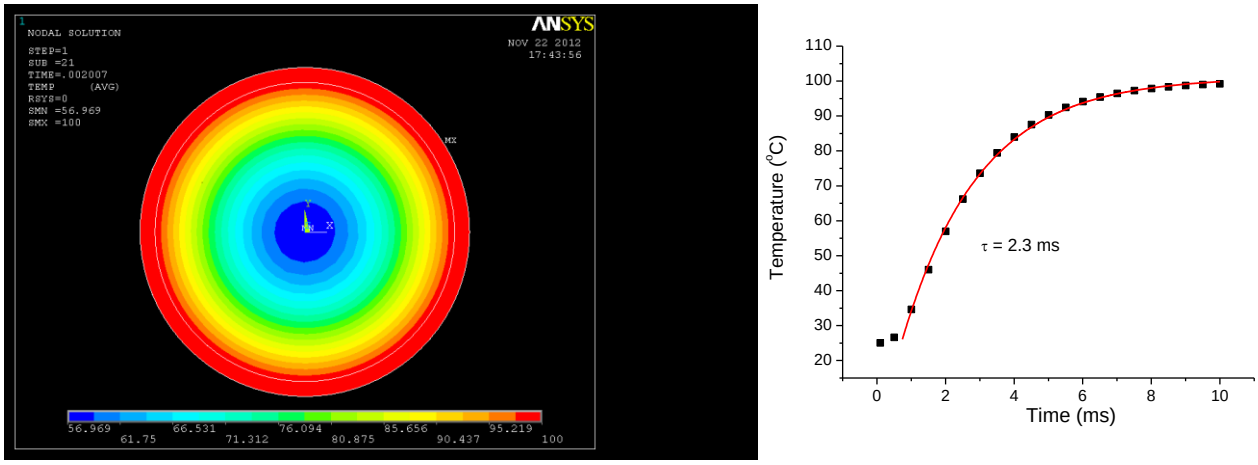

Figure 1: Transient analysis using ANSYS software showing heat transfer from the tube wall into the water inside the tube. (left) Temperature distribution inside the tube at time t=2 ms. (right) temperature of water in the tube centre as function of time. Extracted data shows that the water droplet will warm up with thermal time constant of 2.3ms. It effectively means that in time t= 3*τ = 6.9ms the droplet reaches temperature of the surrounding walls.