

Self-Pumped Coolant Circulation Driven by Thermal Gradient over Microstructures

Nicolas Lobato-Dauzier¹, Anthony J. Genot² and Hiroyuki Fujita¹

¹CIRMM-Institute of Industrial Science, The University of Tokyo,

²LIMMS/CNRS-Institute of Industrial Science, The University of Tokyo

hfujita@iis.u-tokyo.ac.jp

Abstract. We propose a passive cooling device which converts thermal energy into fluid flow via Marangoni effect: a gradient of temperature induces a gradient of surface tension. This in turn triggers a fluid flow at the air/liquid interface which runs transversally to the heat flow. We show how to amplify the global fluid flow thanks to geometrical optimization. We also show how the addition of an asymmetric small wall across the channel reduces the backflow and improves our device.

1. Introduction

Electronic devices produce heat which needs to be dissipated - often with a fluid flow -, which implies the availability of an external supply of energy. The idea behind this work is to use the dissipated heat to propel the cooling fluid. Our aim is to power a micro-pump without applying an external pressure gradient (**Figure 1**), which enables closed loop flows that cycle between hot and cold sources. Recycling the dissipated heat to propel the fluid itself would not only reduce energy consumption, but would boost the autonomy of ambient devices, such as those needed for « the Internet Of Things ». Such devices are very small and therefore a coolant circulation based on MEMS technology is very suitable. At the micrometer scale surface tension effects become very strong, the Marangoni effect – which depends on the variation of surface tension with temperature – is therefore suitable to propel a fluid at such scale.

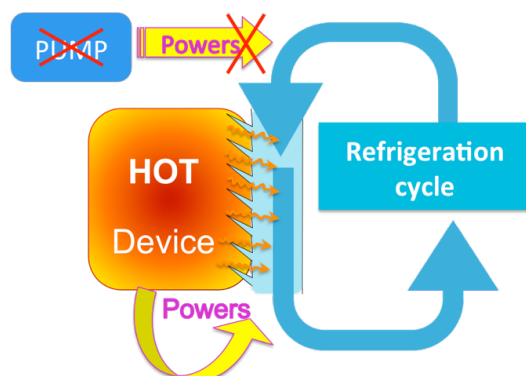


Figure 1. Scheme of the concept: the dissipated heat powers the refrigeration cycle without the need of a pump

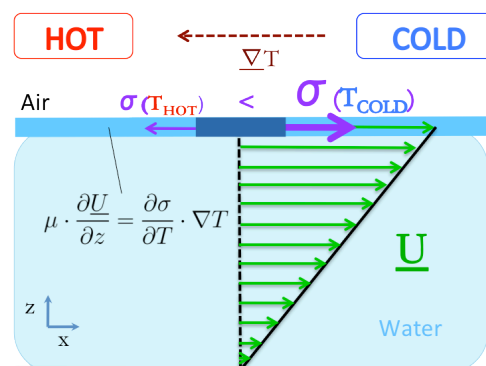


Figure 2. Schematic side view of Marangoni flow: the fluid at the surface is pulled by the cold side

At interface air/fluid, the fluid is pulled stronger on the cold side. For an interface in the xy-plane, this can be expressed by :

$$\mu \cdot \frac{\partial U}{\partial z} = \frac{\partial \sigma}{\partial T} \cdot \nabla T \quad (1)$$

The flow triggered by such effect- also called Marangoni flow- is a couette flow (**Figure 2**). We show how to amplify this effect thanks to geometrical optimization -first developed for autophoretic pumping [1] - which engineers temperature gradients between the heat and cold source. Our proposed device differs from most Marangoni-based pumps, which often considered the effect from a 2 dimensional perspective [2], or applied a longitudinal gradient (**Figure 3A**) of temperature with respect to fluid flow [3]. Although a transverse heat flow (**Figure 3B**) should only create a transverse rotating motion, microstructures allow the fluid to move in the $x > 0$ direction. (**Figure 3C**)

Crucially, because in our device the fluid flow runs transversally to the heat flow, we can decouple the hydrodynamic and thermal scale: this means we can propel fluid over centimetres thanks to the large, but localized thermal gradients that MEMS fabrication enable.

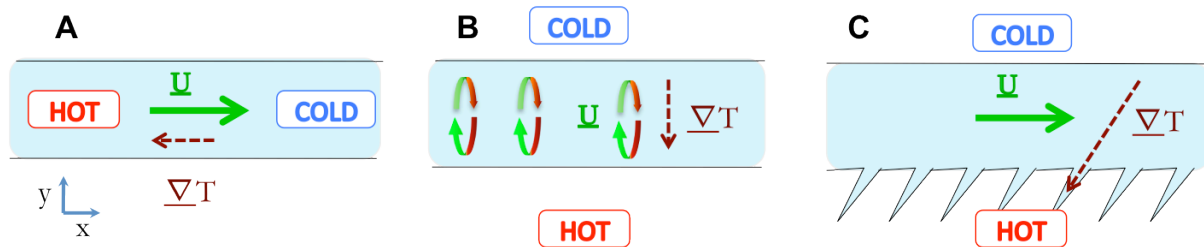


Figure 3. Schematic top view comparison of the Marangoni flow induced by different channels. **A:** Flat channel + parallel gradient / **B:** Flat channel + transverse gradient / **C:** Patterned channel + transverse gradient. Patterning allows the formation of a fluid flow transverse to the original gradient of temperature.

2. Preliminary study and modelization

2.1. Device concept and characteristics

Our device consists on a succession of cavities in which the fluid is pumped out at the surface and pumped in the lower layers. The angle between the cavity and the channel enables the outgoing flow to trigger a surface flow to the right. A wall prevents the pumped-in fluid to come from the same side, which would suppress the global flow in the $x > 0$ direction. (**Figure 5**)

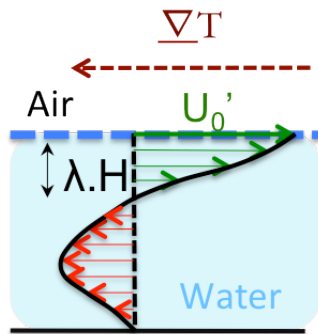


Figure 4. Schematic side view of flow inside a cavity.

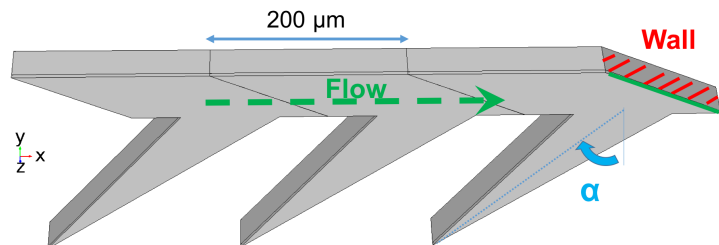


Figure 5. COMSOL model of our device: the red wall stops the backflow from the right, while the fluid on the top can still go over.

2.2. Simulation parameters

We used COMSOL to simulate the flow field inside our device. We only considered diffusive heat transfer, and a Stokes flow. For the heat transfer, we used periodic boundaries for the narrow upper left and right side, set the temperature on the upper wall and modeled the device to be cooled by a heat source. The rest of the boundaries were thermally insulated. For solving the Stokes flow, we used equation (1) as a boundary condition on the top plane, periodic flow condition on the narrow upper left and right side, while considering all other boundaries as “no slip” walls. The fluid chosen is water, and we considered heat source of 65 W/m^2 , which corresponds to a gradient of 1 K/mm for a hot source made of copper for instance.

3. Results and discussion

The simulations show how the micro cavities create a temperature gradient that is not orthogonal to the channel (**Figure 6**). The angle α is therefore the key parameter.

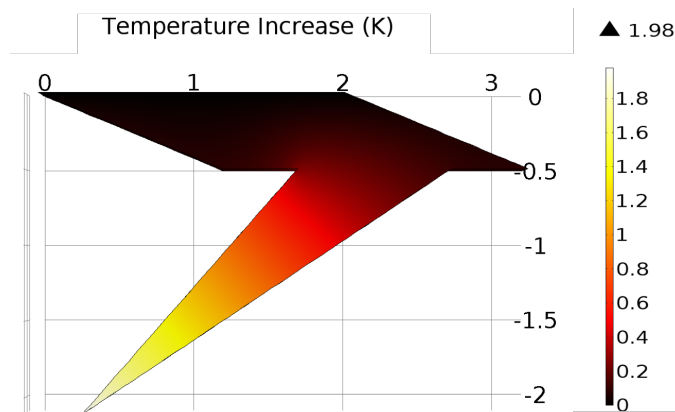


Figure 6. Simulation result for temperature distribution in the channel, for an angle of 50°

As predicted, at the surface the fluid is pumped out of the cavity at a relatively high velocity (**Figure 7**), whereas it is pumped in in lower layers (**Figure 8**). The addition of the wall works properly : it only allows the top layers to pass while preventing backflow. Preventing the backflow from the right side allows us to have a wide opening for the cavity, which can propel more fluid then. However the cost is quite high, as having such a wall decreases the velocity.

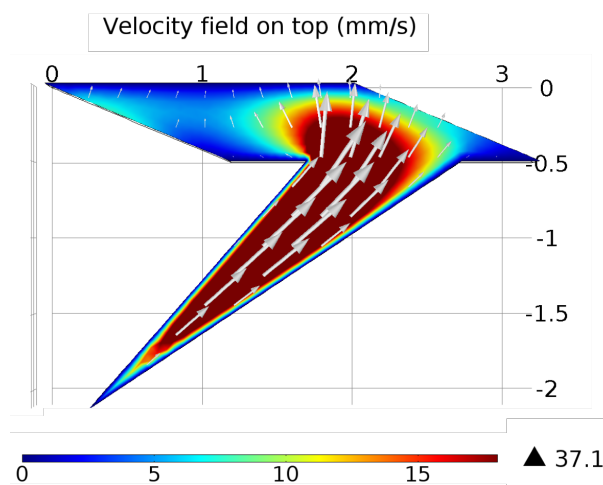


Figure 7. Simulation result for the velocity field on top : the fluid is pumped out at high velocity

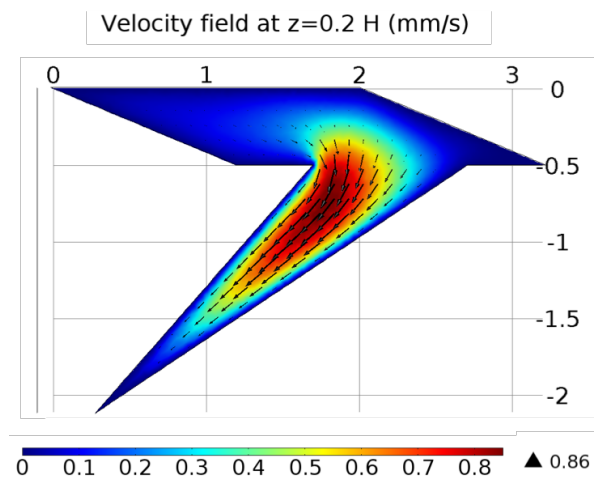


Figure 8. Simulation result for the velocity field near the bottom: the fluid is pumped in, mainly from the left side, at a lower velocity

The variation of the angle of the cavity reveals a trade-off relationship: high angles are more efficient to propel the fluid in the right direction. However, when the angle increases the cross section of the cavity decreases and therefore the volumic flow rate also decreases. There is therefore an optimal angle which measures around 50° (**Figure 9**)

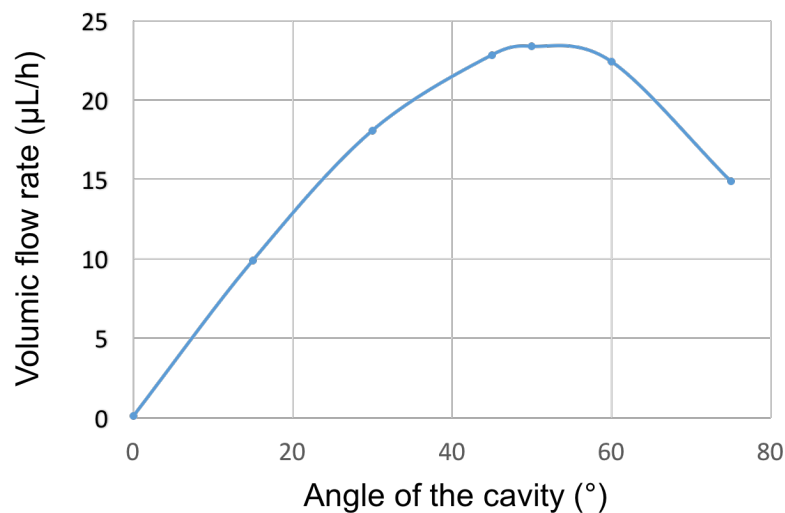


Figure 9. Evolution of the volumic flow rate with the variation of the cavity angle. There is an optimal angle that maximizes the flow rate around 50°

4. Conclusion

We showed that thanks to micro cavities on a channel walls, it was possible to use Marangoni effect to propel a fluid in a direction orthogonal to the temperature gradient with flow rates that however small they are, correspond to what is used in some microfluidic devices. As we can decouple the hydrodynamic and thermal scale this device is suited for propelling a fluid over distances that can reach the centimeter while having the benefits of the large thermal gradients that MEMS fabrication allows.

5. References

- [1] S. Michelin et al. , *Soft Matter*, 2015,**11**, 5804-5811
- [2] C. Maggi, et al. *Nature Communications*, 2015, **6**, 7855
- [3] Michael Grunze, *Sciencemag* 1999, **283**, 5398