

Heat Transfer in a Trapezoidal Micro-channel with Laminar Fluid Flow

Shantanu Shukla
Department of Mechanical Engineering, IIT Bombay

Synopsis

A laminar flow was simulated through a trapezoidal micro-channel of specified dimensions with a semi-circular and rectangular cross-section using OpenFOAM v2012. With the consideration of a constant wall heat flux from the walls of the channel the heat transfer characteristics were studied assuming a fluid of Prandtl number 6.13 flowing through the channel. The variation in the Nusselt number with respect to the Reynolds number was observed for both the geometries along with a comparison of wall temperatures and centerline (or axis) temperatures. Comparison of the Nusselt numbers of the trapezoidal geometry with a straight channel gives us an estimate of the effectiveness of heat transfer enhancement due to the modified geometry. The reference [1](*Laminar Flow and Heat transfer in a periodic trapezoidal channel with semi-circular cross-section, Paul .E Geyer et al.*) around which this Migration Project revolves, uses Ansys CFX 13 for the simulations.

1 Introduction

There has been a recent drive in the miniaturisation of heat transfer devices including solar heat collectors, cooling in electronics, microchannel processing and creating a compact heat exchanger device. At a microscale, the Reynold's numbers of operation are too low to benefit from turbulence effects and hence geometric properties of a microchannel are dominant in affecting the rate of heat transfer. Thus the enhancement of heat transfer in a laminar flow can be achieved through making the flow follow a tortuous path. Previous works have dealt with serpentine channels with various cross-sections and geometries to obtain high heat transfer rates. However, serpentine channels have a poor stackability when it comes to affixing these channels over thin plates. Hence, a better alternative for this is a channel with a trapezoidal axial path, which is analysed in this work. Instead of a channel with semi-circular cross-section, we work with a rectangular cross-section and attempt to obtain similar results.

2 Governing Equations and Models

The SIMPLE algorithm is used for obtaining pressure and velocity in the channel. The governing equations are the continuity and momentum equation which are stated as below:

$$\frac{\partial(u)}{\partial t} + \nabla \cdot (u \otimes u) = -\nabla p + \rho g \quad (i)$$

$$\nabla \cdot u = 0 \quad (ii)$$

The equations (i) and (ii) represent the momentum and continuity equations respectively. The momentum equation is discretized as mentioned in reference[2] and the velocity corrector and flux corrector terms are obtained which then lead to the pressure equation. In the momentum equation, the pressure and the gravity terms are separated as in equation (iii) which leads to a p_{rgh} term.

$$\begin{aligned} -\nabla p + \rho g &= -\nabla(p_{rgh} + \rho g \cdot r) - \rho g \\ \text{where } p_{rgh} &= p - \rho g \cdot r \\ -\nabla p + \rho g &= -\nabla p_{rgh} - (g \cdot r)\nabla \rho - \rho g + \rho g \\ -\nabla p + \rho g &= -\nabla p_{rgh} - (g \cdot r)\nabla \rho \quad \dots (iii) \end{aligned}$$

The energy type used is the internal energy, which is denoted as **sensibleInternalEnergy** in OpenFOAM and is present in the **thermophysicalProperties** file in the **constant** folder. The governing equation for the same is as given below.

$$\frac{\partial(\rho e)}{\partial t} + \nabla \cdot (\rho u e) + \frac{\partial(\rho K)}{\partial t} + \nabla \cdot (\rho u K) + \nabla \cdot (p u) = \nabla \cdot (\alpha_{eff} \nabla e) + \rho u \cdot g$$

where $K = |u|^2/2$ the kinetic energy per unit mass

$e =$ internal energy per unit mass

α_{eff}/ρ is the effective thermal diffusivity which is the sum of laminar and turbulent thermal diffusivities.

The thermophysical properties and other parameters used in the simulation were as follows :

WorkingFluid – Water

$C_p = 4182 J/KgK$ – specific heat capacity

$\mu = 1.7894 \times 10^{-5}$ – dynamic viscosity

$\rho = 1000 kg/m^3$ – Density

$Pr = 6.13$ – Prandtl number

$D = 10^{-6} m$ – Hydraulic diameter of channel

The Nusselt number after the simulation is calculated as follows:

$$Nu = \frac{\frac{\partial T}{\partial x}|_{wall}}{\left(\frac{T_{bulk} - T_{inf}}{D}\right)}$$

$\frac{\partial T}{\partial x}|_{wall}$ is evaluated in the post processing part. The temperature profile is obtained along the x axis with a resolution of 100 points. The gradient at the wall is evaluated from this profile in a discretized manner. $\frac{\partial T}{\partial x} \approx \frac{\Delta T}{\Delta x}$

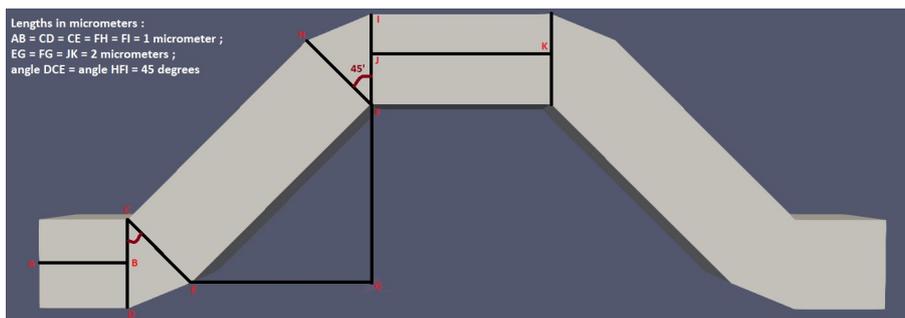
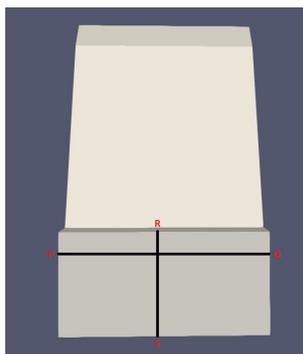


Figure 1: Side view of the channel

Figure 2: Front view of the channel; $PQ = 2\mu m$ $RS = 1\mu m$

3 Simulation Procedure

3.1 Geometry and Mesh

The trapezoidal channel has the dimensions as shown. The horizontal entrance and exit lengths are $1\mu m$ each. The horizontal top portion of the channel is $2\mu m$. The inclined portions of the channel are $2\mu m$ in length and rise. The inclination after the entrance is provided at a 45 degree angle without any curvature.

The meshing for the rectangular cross-section channel was done using 'blockMeshDict' in OpenFOAM itself. A total of 70,000 cells have been used to mesh the channel, with the size of cells reducing by 10 times near the corners and an otherwise uniform grading

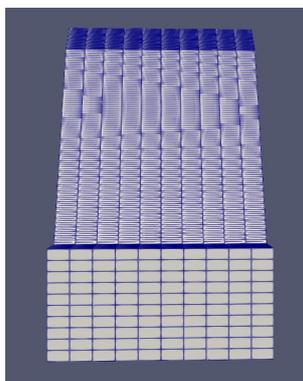


Figure 3: Front view of the meshed channel

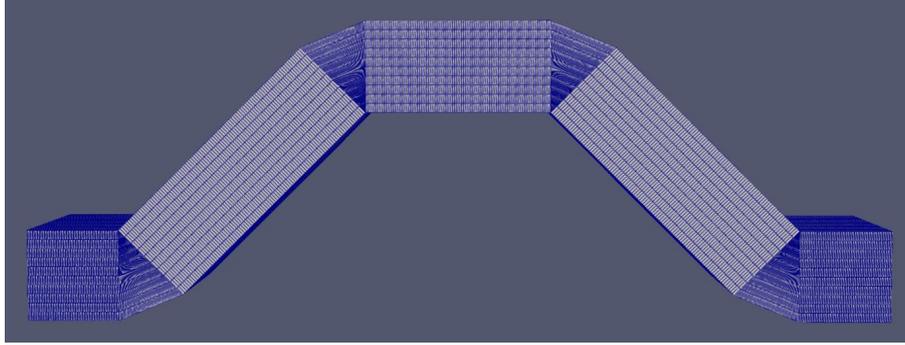


Figure 4: Side view of the meshed channel

along the x and y directions.

3.2 Initial and Boundary Conditions

Velocity and Pressure boundary conditions - Velocity boundary conditions include no slip boundary condition at the wall and a constant inlet and outlet velocity which is specified by us. The inlet and outlet velocities lie within a range of 0.001m/s to 1m/s and the results are studied for different velocities. It is this velocity that determines the Reynolds number. Pressure at the inlet and outlet faces is as calculated using the `$internalField` functionality. A **zeroGradient** pressure boundary condition is present at the walls.

Temperature boundary conditions - At the channel walls we consider a constant heat flux of $q = 1000W/m^2$ which is roughly $10^{-9}W/\mu m^2$. The inlet and outlet operate at a **zeroGradient** temperature condition.

Note - We are considering this channel to be a part of a series of such channels used in a heat exchanger. Depending on the velocity we specify, OpenFOAM adjusts to the corresponding pressure drop.

Initial Conditions - Zero velocity and pressure initial conditions and a uniform initial temperature of 300K .

The inlet and outlet are declared as **type - patch**. The working fluid is **Water**.

3.3 Solver

We use the **buoyantSimpleFoam** solver to simulate the results. The heat transfer takes place through convection from the channel walls to the fluid. Since we are working in the steady-state laminar regime, we are using the SIMPLE solving algorithm in **buoyantSimpleFoam**. Along with the **p**, **U**, and **T** files, a **p_rgh** file is also present which takes into account the hydrostatic contribution for Pressure.

4 Results and Discussions

The above system with a rectangular cross-section was simulated for velocities 0.001m/s through 0.6 m/s. The general velocity profile obtained was as given in Figure 5. It shows the variation of velocity through the volume of the channel and the graph is the velocity profile along a line parallel to the x-axis, in the center plane of the top horizontal part of the channel.

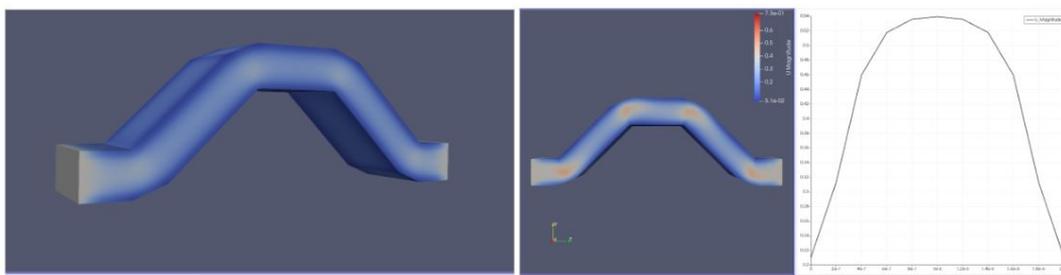


Figure 5: $V = 0.4m/s$ velocity profile

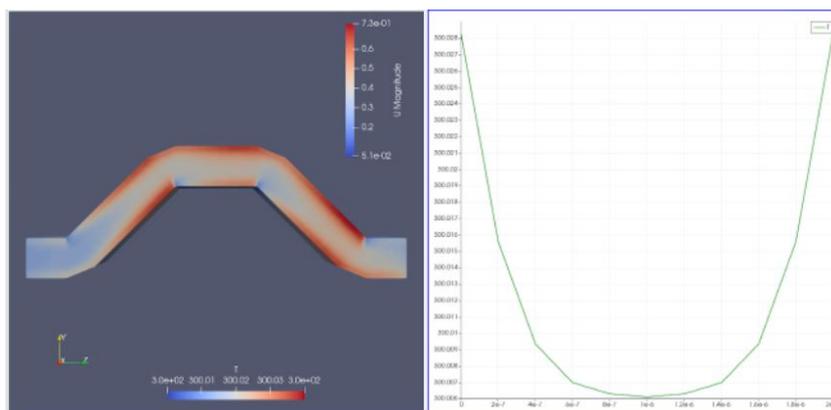


Figure 6: $V = 0.4m/s$, Temperature profile

The temperature variation at the steady state in the channel is shown in Figure 6. The graph in figure 6 shows the temperature variation along a line parallel to the x-axis passing through the midpoint of the upper horizontal section of the channel. Figure 6 shows a micro-variation in the the temperature distribution across the channel. Also note that the temperature rise near the walls is very small, since the area of cross-section is small, and the heat influx is currently considered to $10W/\mu m^2$. The variation of temperature over the cross-section of the channel can be observed in the contour plot in Figure 7.

Such temperature variations were obtained for different values of flow velocities, which

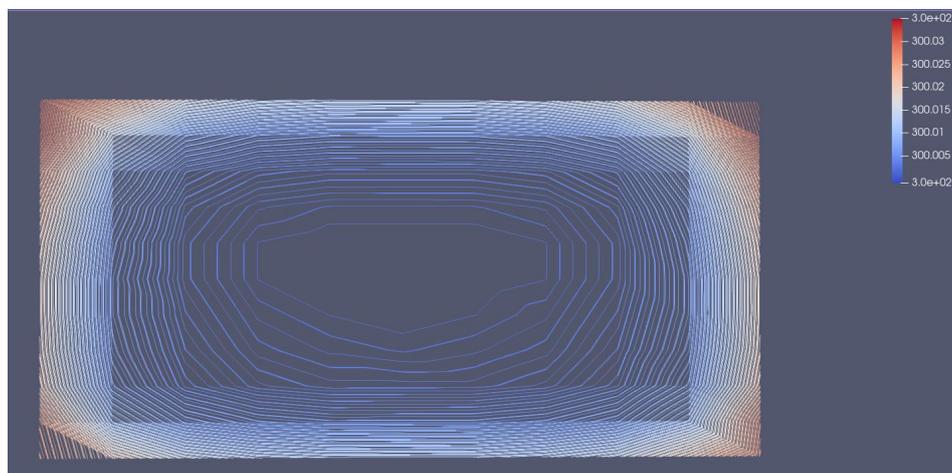


Figure 7: Temperature Contours

velocity (m/s)	wall temperature(K)	minimum temperature(K)	Nu	Re
0.001	300.11	300.072	5.128	0.055
0.002	300.11	300.071	5.128	0.111
0.003	300.11	300.071	5.128	0.167
0.004	300.11	300.071	5.128	0.223
0.005	300.109	300.069	5	0.279
0.006	300.109	300.068	4.878	0.335
0.007	300.108	300.067	4.878	0.391
0.01	300.106	300.064	4.761	0.558
0.1	300.058	300.021	5.405	5.588
0.2	300.05	300.016	5.882	11.17
0.4	300.028	300.006	9.09	22.35
0.6	300.024	300.004	10	32.53

Figure 8: Variation of Temperatures and Nusselt numbers with velocity

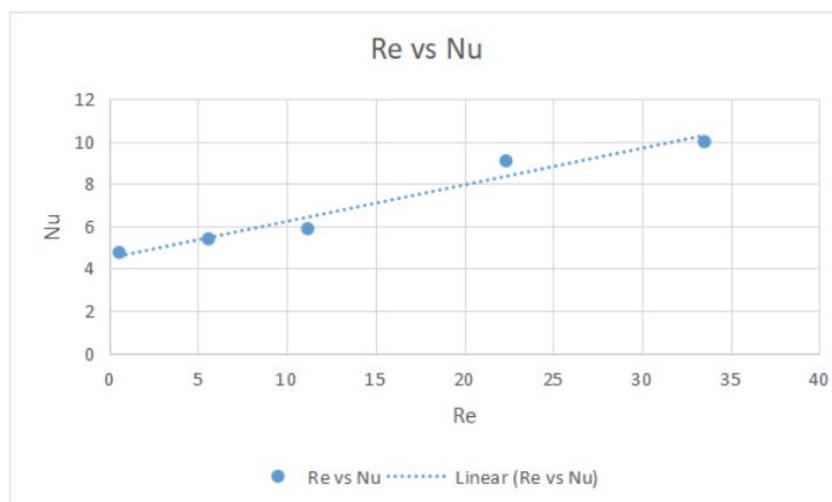


Figure 9: Re vs Nu for a rectangular channel

means different Reynolds numbers and an effective tabulation was made as follows.

Based on the above table, we observe that initially the value of the Nusselt number remains more or less constant for extremely low values of velocities. Beyond a certain velocity, as the velocity is increased, a significant change starts originating with a high rise in Nusselt number with velocity. Figure 9 shows that as the Reynolds number increases, the Nusselt number also increases along with decrease in wall temperature.

The effectiveness of the channel is proportional to the Nusselt number that has been calculated we see that with the increase in Reynolds number, the effectiveness of the channel would increase. This is however, provided that we continue to operate in the laminar regime.

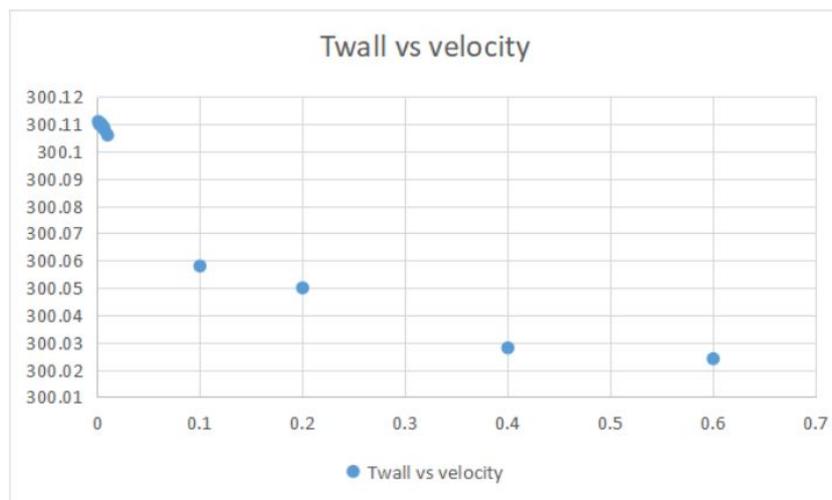


Figure 10: Wall Temperature vs velocity

References

- [1] Laminar Flow and Heat transfer in a periodic trapezoidal channel with semi-circular cross-section, Paul .E Geyer et al.
- [2] OpenFoam User Guide v2112

DISCLAIMER: This project reproduces the results from an existing work, which has been acknowledged in the report. Any query related to the original work should not be directed to the contributor of this project.