BMT-3076 Microfluidics

E3: FEM modeling pressure driven flows 2

Numbers of microfluidic devices have long straight channels with constant cross-section. Flow through such channels is an important flow phenomenon at microscale. It can be studied with an equation below.

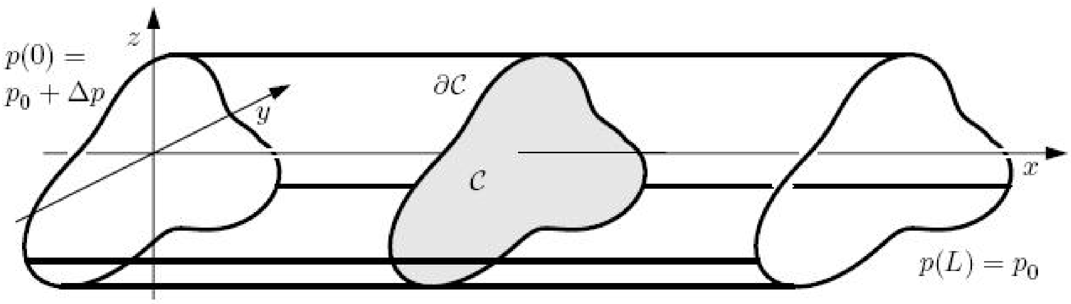

Figure 1 The Poisseuille flow in a channel with constant cross-section

$$\left[\partial_y^2 + \partial_z^2\right] v_x(y,z) = -\frac{\Delta p}{\eta L}, \text{ for } (y,z) \in C \qquad (1)$$

$$v_x(y,z) = 0, \text{ for } (y,z) \in \partial C$$

Study pressure driven flow in straight microchannels with rectangular cross-section. Use Comsol Multiphysics to solve problem numerically. Use 2D modeling space, stationary study type and Mathematics/Classical PDEs/Poisson’s equation (poeq) as physics. Use of parametric sweep is recommended in optimization.

1. Solve flow rate in the case of very tight mesh (100 elements (n) in the edge)
2. Optimize mesh density (accuracy vs. simulation time) using mapped mesh (square elements). What is the minimum number of elements (n) in the edge to have less than 0.01 % error in flow rate compared to very tight mesh (n = 100)?
3. Simulate hydraulic resistance as a function of height of the channel from 10 µm to 400 µm using optimized mesh.

Poisson equations:

$$\left[\partial_y^2 + \partial_z^2\right] v_x(y,z) = -\frac{\Delta p}{\eta L}, \text{ for } (y,z) \in C$$

$$v_x(y,z) = 0, \text{ for } (y,z) \in \partial C$$

Initial values
w = h = 200 µm
L = 250 mm
η = 0.001 Pa s
ρ = 1000 kg / m^3
Δp = 3 kPa

Hints:
Define parameters (Global definitions / parameters)
Create geometry using parameters
Set source term f as Δp / (L*η) in Poisson's Equation 1 as per formula (1) above.
Add Dirichlet boundary condition to all boundaries.
In Mesh add Distribution (select all boundaries, set number of elements) and Mapped
Use Results / Derived values / Surface integration to get flow rate.

In part 2:
Create probe: Model1/Definitions/Probes/Domain probe, type Integral
Add parametric sweep for number of elements from 1 to 2
Run model
After that in Job configurations/ Parametric1:

- Select defined by study step: Parametric sweep
- Create stop condition to stop increasing n once the flow rate is closer than 0.01% to "n=100" case. Stop condition stops sweeping when expression < 0. In expression variable **mod1.dom1** (or **geom1.dom1** etc, depending your model) gives the flow rate at the latest element number.
- Change parametric sweep to go from 1 to 100

In part 3:
Remove the stop condition.
Change parametric sweep to modify the height of the channel. Use 10 µm as step.
Copy data to excel or Matlab to calculate and plot hydraulic resistance. Use logarithmic scale for y.

The results (flow rate with n=100, optimized n, R-hyd plot) should be returned to the assistant using Moodle.