## Goal

The goal of this research is to model dissolution of a material from a cavity in a microchannel, which later can be applied toward the development of point-of-care diagnostic devices. This research is done in conjunction with Elena Garcia, a graduate student in Paul Yager's group.

## Femlab Approach

To model the cavity and the dissolution in Femlab, the dimension of the model is specified as shown below:

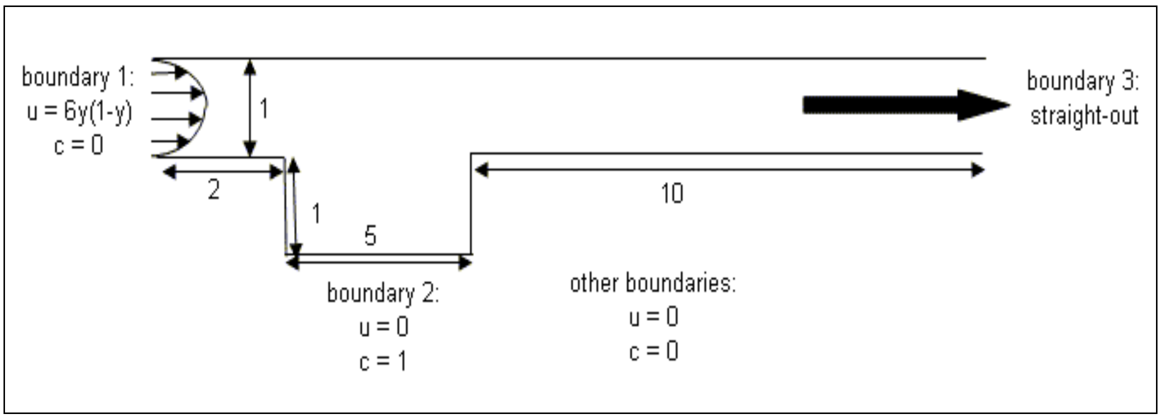



**Figure 1. The sketch of the model used in Femlab. Note that the number used to specify the boundary in this paper is not the same as the one used in Femlab.**

Navier-Stokes and Convection-Diffusion modes were used to approach this problem.

**Equation 1**

$$\frac{\partial u'}{\partial t'} + \mathrm{Re}\,\underline{u}'\cdot\nabla'\underline{u}' = -\nabla' p' + \nabla'^2\,\underline{u}'$$

Dimensionless Navier-Stokes mode.

**Equation 2**

$$Sc\frac{\partial c'}{\partial t'} + \underline{u}'\cdot\nabla' c' = \frac{1}{Pe}\nabla'^2\,c'$$

Dimensionless Convection-Diffusion mode

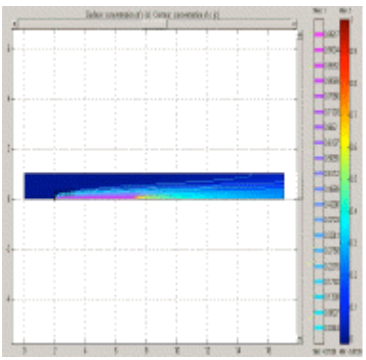
The Navier-Stokes mode was used to calculate the flow problem. Here, a hyperbolic velocity profile with a fully developed flow was introduced at the channel entrance and a straight-out boundary at the channel exit. Then, the Convection-Diffusion was performed to specify the concentration. Concentration of 1 was specified at the bottom of the cavity (boundary 2), 0 i.e. constant flux for other boundaries, and a straight-out for the exit (boundary 3).

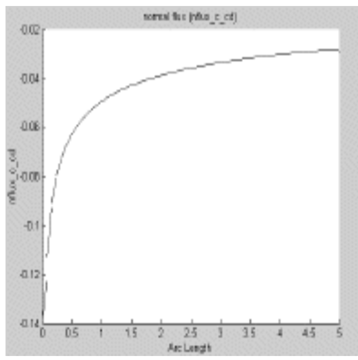
The subdomain was set to have Re = 10, μ = 1 for Navier-Stokes mode and Pe = 100 for initial run. Other runs were also performed at various Peclet number and various L/H ratio to see the behavior of each condition.

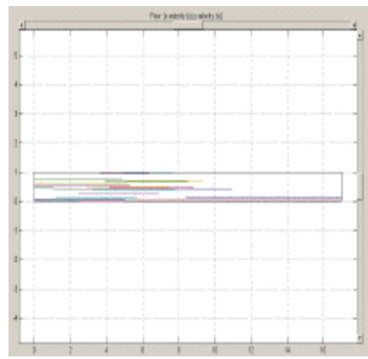
A concentration contour plot, a velocity flow line plot, and a flux plot at the boundary 2, i.e. the bottom of cavity, were made for each case. The concentration and velocity flow line plots showed the concentration and velocity profiles through out the channel and the cavity. The flux plot at boundary 2 would help in determining the shape of the optimum cavity that would be modeled later.

## Results

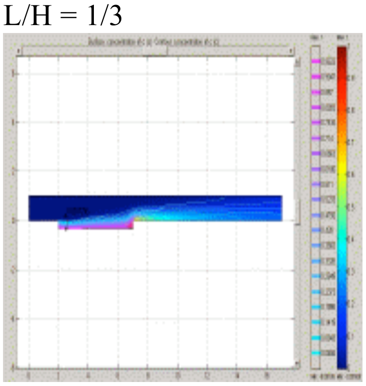
The results at different layer of cavity are shown below:
L/H = 0

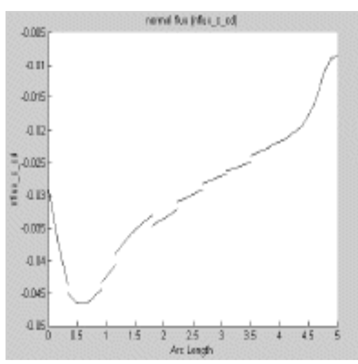


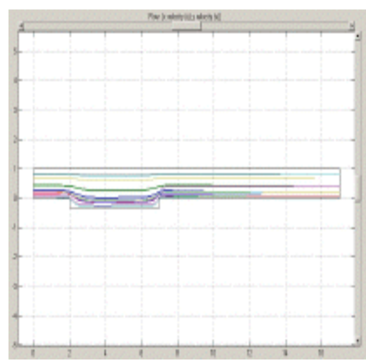


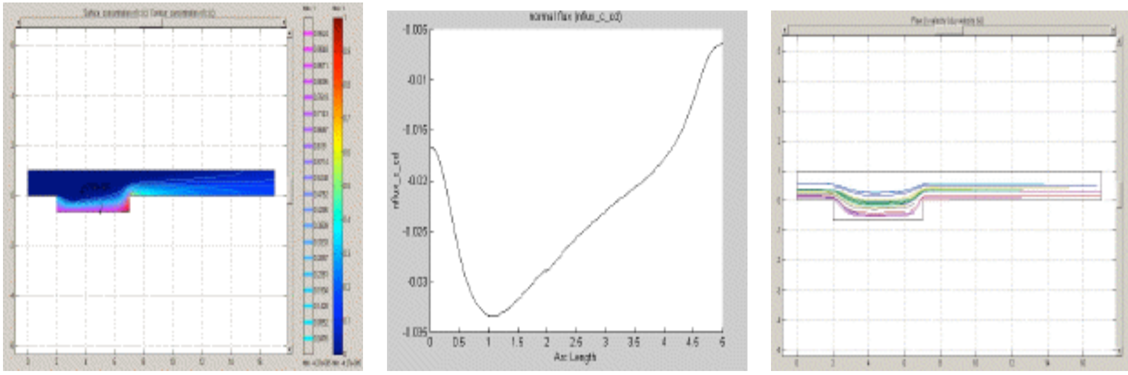
L/H = 1/3



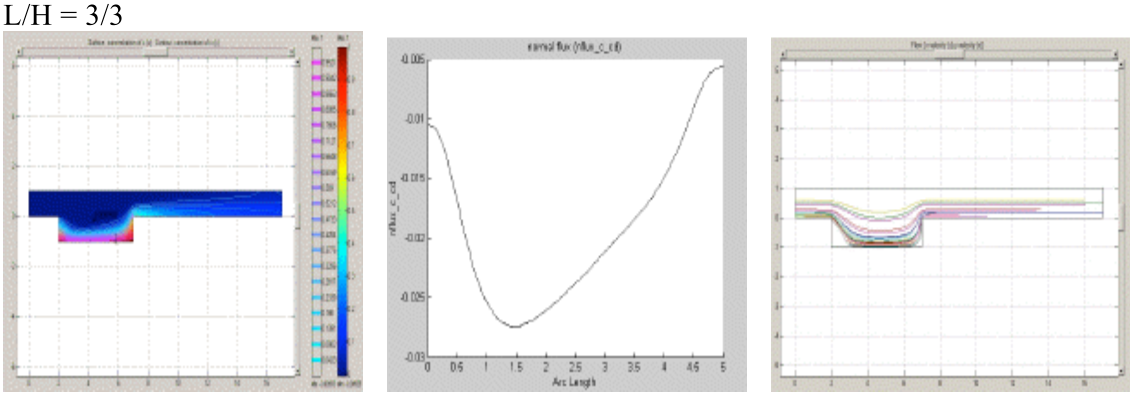


L/H = 2/3

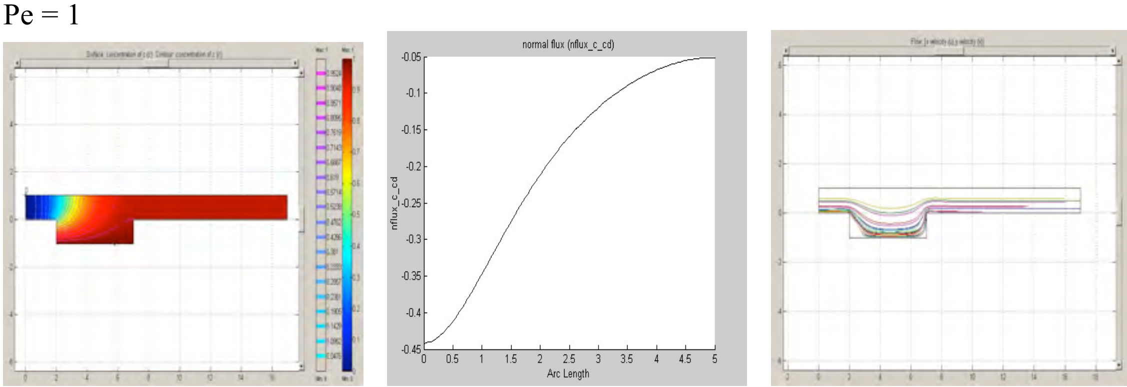



L/H = 3/3

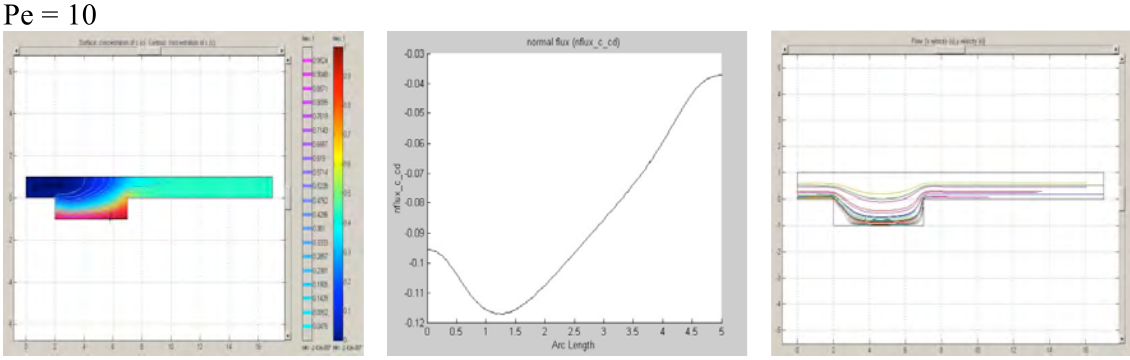



The results of various Pe at Re = 10, $\eta = 1$ are shown below:

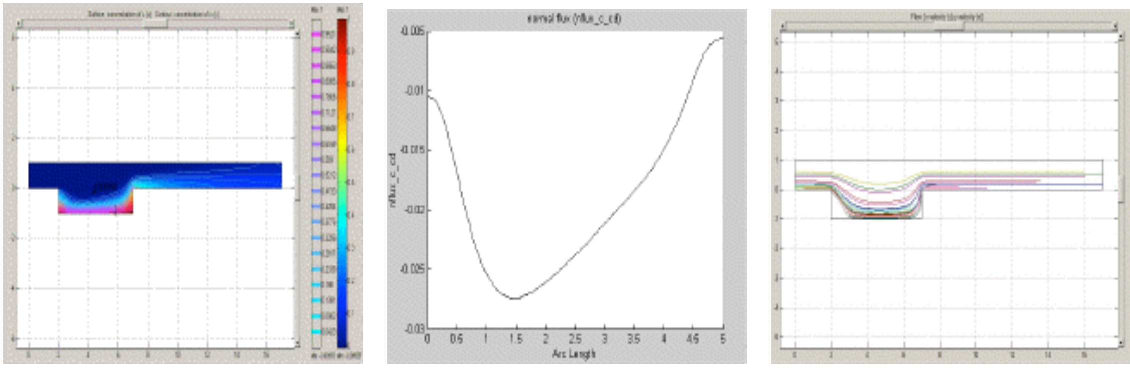
Pe = 1



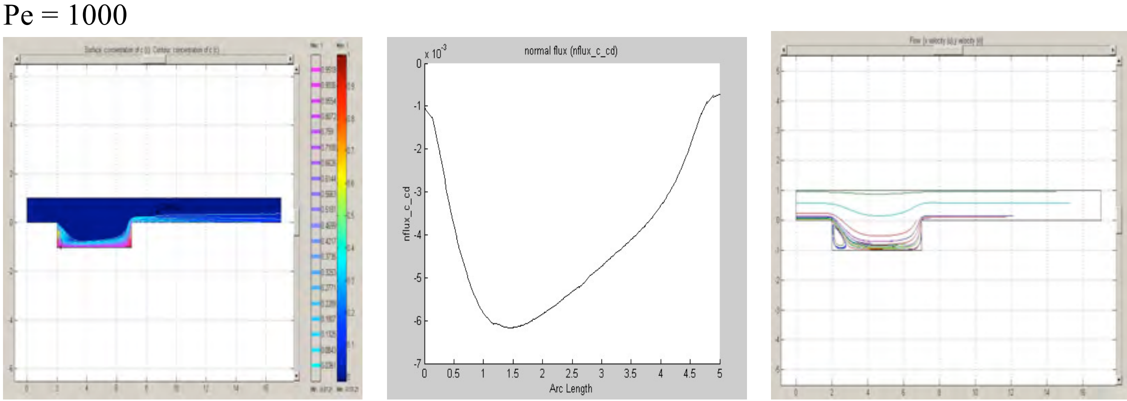

Pe = 10



Pe = 100




Pe = 1000




## Quasi-Static Approximation

Quasi-Static approximation was done to model the shape of cavity so that all materials at boundary 2 were able to be dissolved by the fluid flow. This approach was performed after equation 3 and 4 as shown below:

**Equation 3**

$$\frac{amount\ removed}{w} = \Delta x \cdot height \cdot c = flux \cdot \Delta x \cdot \Delta t$$

where amount removed = amount of materials being removed
w = width/thickness of cavity
height = new height of cavity
flux = flux at bottom of cavity

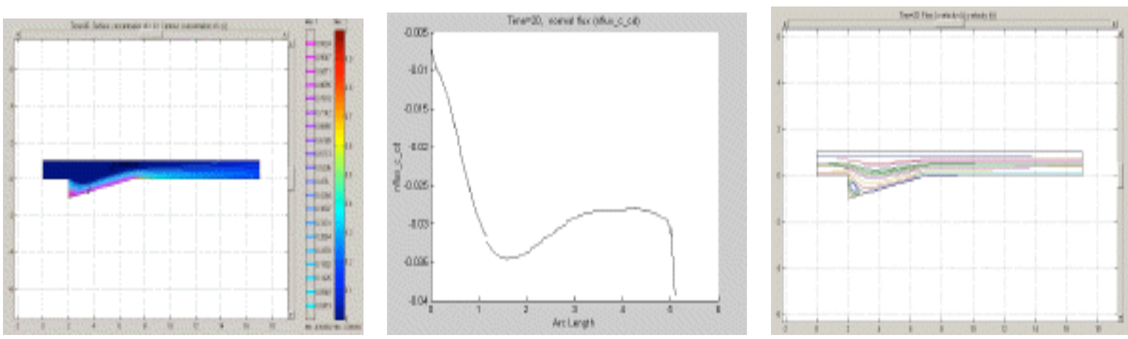
convert equation 3 to dimensionless in Femlab:

**Equation 4**

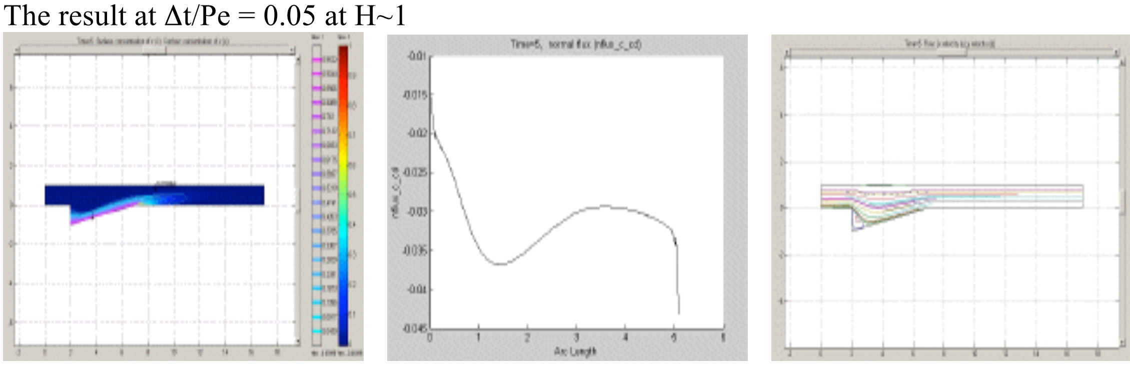
$$height' = \frac{\Delta t'}{Pe} \cdot flux'$$

Starting using a model with ratio of L/H = 0, a boundary integration of the concentration at boundary 2 was obtained. This number, i.e. the flux, and a ratio of Δt/Pe then are used in equation 4 to calculate the height of the cavity.
A triangle shaped cavity with a curved hypotenuse was chosen to approximate the shape of the cavity due to its ability to dissolve more materials. The three plots of concentration, velocity flow line and flux at boundary 2 were obtained after running the simulation. The same procedure was done until reaching the original height of the cavity; this case H=1.

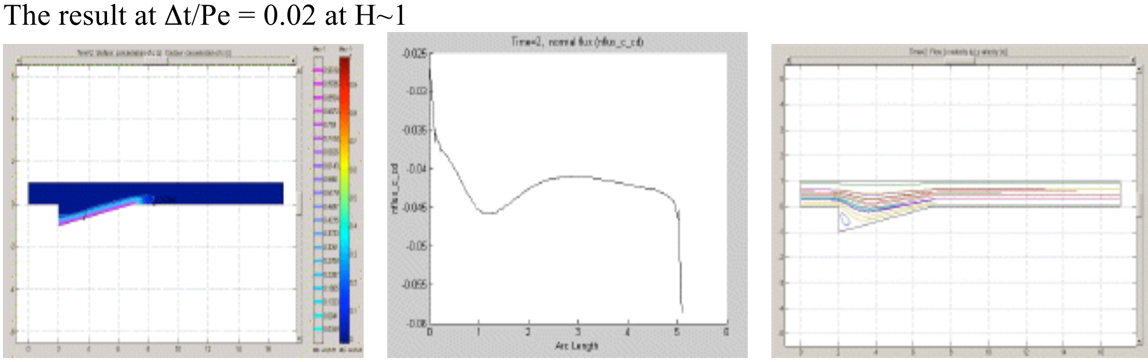
The result at Δt/Pe = 0.2 at H~1



The result at Δt/Pe = 0.05 at H~1



The result at Δt/Pe = 0.02 at H~1

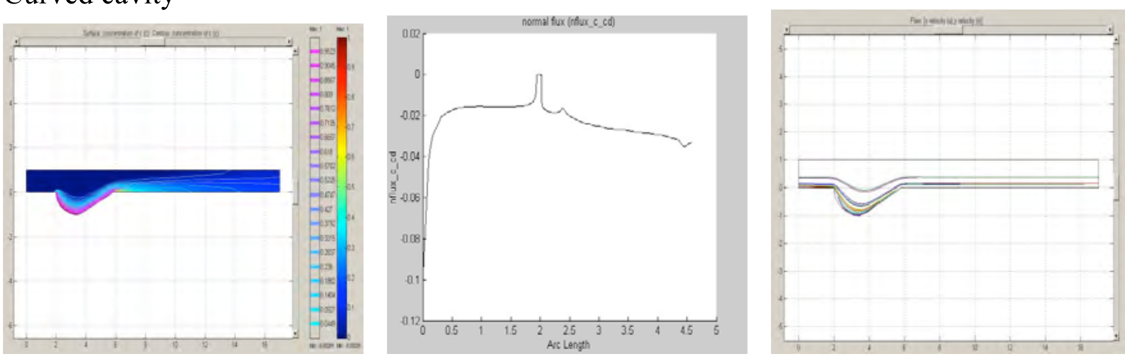


To test the validity of Quasi-Static approach, different Δt/Pe ratio was used and compared the results at H ~ 1 of each Δt/Pe ratio. Here, the Quasi-Static was proved to be a good approach to model the shape of the cavity because all results at H ~ 1 had similar behavior.

## Better Shape

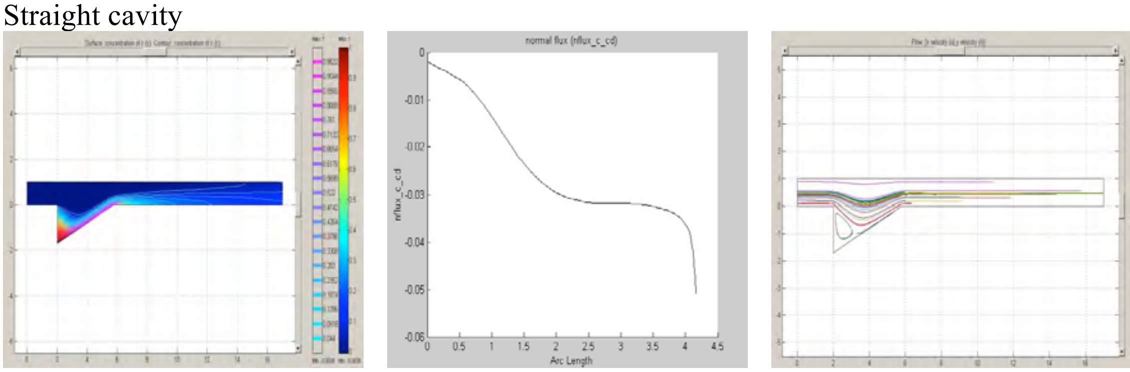
To eliminate the recirculation at the corner of the cavity, the shape of the cavity needed to be adjusted. A curved cavity was a better choice to eliminate this recirculation, but this model was more difficult to draw, especially if it would be done in 3-dimension.

Curved cavity




Note: a better/smooth drawing of the cavity would make a better flux plot.

Straight cavity




## Acknowledgements

- Elena Garcia
- Prof. Paul Yager
- Finlayson's and Yager's research groups