

Department of Microelectronics and Computer Science
Technical University of Lodz

Biochip

(ver.2)

Krzysztof Slusarczyk, M.Sc.
kms@dmcs.p.lodz.pl

Lodz, July 2002

Problem description: biochips are modern devices widely used in biology, medicine, chemistry and biochemistry. They are a part of more complicated system, which usually contains of a biochip itself (which consists of capillary or array of capillaries), set of sensors and a computer, which collects and computes all data. Biochips are more often made in a glass (most common used is Pyrex® glass), or in silicon wafer.

In old biochips fluid was moved by pressure difference between inlet and outlet, but in modern biochips fluid is driven due to electrokinetic phenomena: electroosmosis and electrophoresis.

Electrophoresis – The basis for electrophoresis is the differential migration of the charged species ions relative to the carrier molecules under the application of the external field. The differential migration is preliminary an effect of the difference in the net charge between the solvent and solution ions. In the mixture containing more than one kind of ion, velocity of each charged species depends on its mobility, so concentration of species can be spot during this phenomenon.

Electroosmosis – Electroosmosis is a macroscopic phenomenon involving the pumping of a fluid through a channel under the application of the field. The charge may either be due to the property of the wall or by adsorption of the charged species from the buffer. In the presence of an electrolyte the surface charge density induces the formation of the double layer in the fluid by attracting oppositely charged ions from the electrolyte to the immediate vicinity of the wall. The application of the field exerts a force on the fluid, which is initially felt within the double layer. As a result the fluid in the near vicinity of the wall starts to move. Due to the viscous forces the fluid in the center of the channel is also accelerated until the net velocity gradient in the radial direction is zero and the whole fluid in the channel moves at a constant velocity.

Phenomena described above have complicated nature, so some assumption must be taken into the consideration while equations are being derived. Electroosmosis can be easily described under the following assumptions:

- *Dilute carrier*: the carrier fluid is assumed to be electroneutral everywhere, except within the double layer. The charge distribution is therefore confined to a small region within the double layer
- *Dilute solution*: the carrier fluid is assumed to be predominant species in the mixture. That is, the mass fraction of the charged species is assumed negligible to the fraction of carrier. This implication of the dilute assumption is that the solute species does not affect the material properties of the mixture. This allows the species transport equation to be decoupled from the momentum equation.
- *Individual species do not affect each other*: this assumption is used in describing the flux terms. The flux of one species does not depend on the fluxes of the other species (but that is true only in dilute solution).
- *No chemical reactions*: the charged species are assumed to be fully ionized in the mixture.

The dilute solution allows the density of the mixture to be assumed constant and equal to the density of the carrier. This reduces the problem to the incompressible form. The decoupling of the species equation from the momentum equation also implies that the species conservation condition is not relevant and

can be discarded.

Capillary – is a long (55 to 100cm) glass pipe of small diameter (50 or 75um); in biochips capillary is much shorter (average length is about 3 to 6cm), has a rectangular cross-cut and the width of 50um.

In this tutorial we will be analyzing a very simple model of biochip. The tutorial is divided into three parts. In the first one you will learn how to prepare and run EOS simulation, in the second one – how to simulate EPH process. The third part is a project, you will run a two-step simulation of cross-like biochip.

Part One

EOS Simulation

1. Generating the Microchannel

Our structure is a simple 2D model of a microchannel. In CFD – GEOM you will draw a simple rectangle (1mm by 50um) and set a structured mesh to the structure.

Start a new project in CFD – GEOM.

In *Geometry* tab chose *Point Creation* (Fig. 1), on the left side of the panel appear three type-in fields for X, Y and Z numbers (Fig. 2). Create four points: (0; 0), (1e-3; 0), (1e-3; 5e-5), and (0; 5e-5).

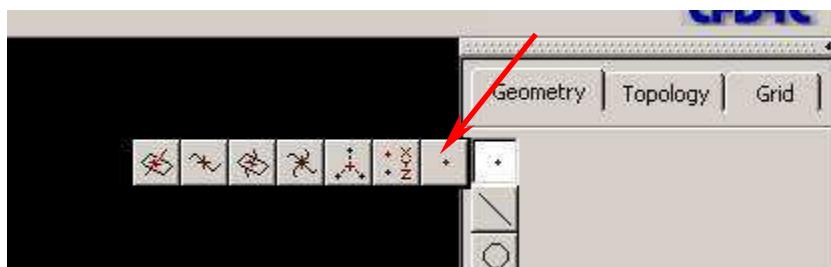


Fig. 1. Point Creation Tab

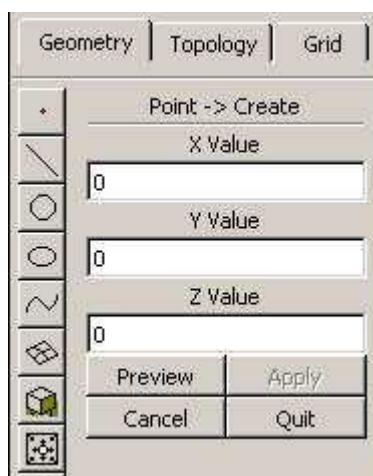


Fig. 2. The view of type-in fields for X, Y, Z coordinates

To connect all points with a line, go to *Line Creation* tab (Fig. 3) and choose a

Polyline. Aim each point with a mouse cursor and click LB to pick it, after connecting all points click MB to accept the creation of the line. You should obtain results similar to the one shown in Fig. 4.

Do not forget to save your work.

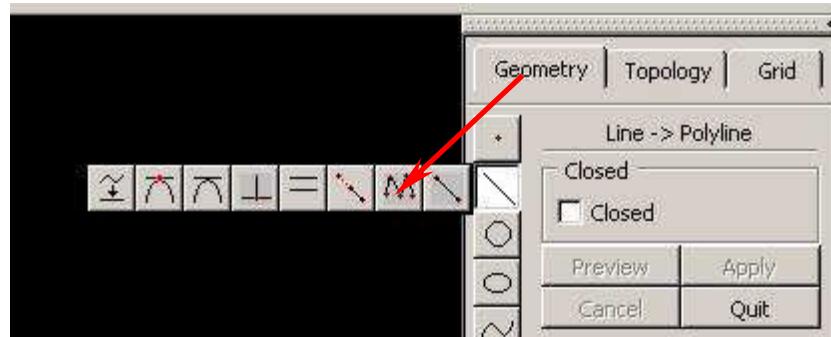


Fig. 3. Line Creation tab (the Polyline option is pointed)



Fig. 4. A model of a microchannel created in CFD - GEOM

2. Adding a Grid

The next step is setting a grid to your model. Switch into *Grid Generation* tab, and in the *Structured Edge Options* select *Create Structured Edge* button (Fig. 5).

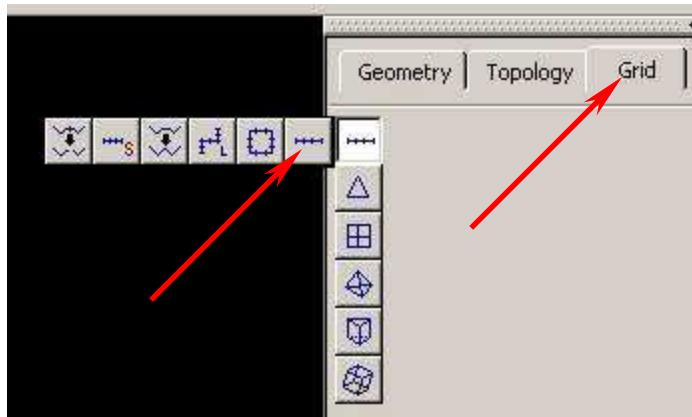


Fig. 5. Create Structured Edge button

To shorten the simulation time you use structured grid. The most optimal grid for CFD - ACEU solver is squared one, that is the reason, why in your model you are going to use structured mesh. To shorten the simulation time you also use one trick – you are going to define mesh in such a way, that the simulation will be nearly 1D. To do this, for the left and the right wall set only 2 grid points, and for the upper and the lower wall set 21 grid points. In that way you create square mesh with walls of 50um.

To set the appropriate number of grid points, enter its number in the *Number of Points* type-in field (Fig. 6) and choose the line you want to set grid points to. After that, you have to apply changes by clicking MB or *Apply* button.

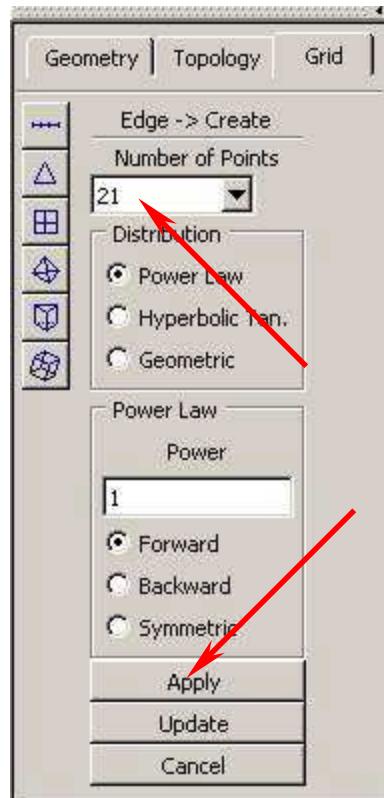


Fig. 6. The view of Edge Creation panel

3. Creating a 2D Block

The next step is to create Structured Face. Go to Structured Face Options and choose Create Structured Face (Fig. 7).

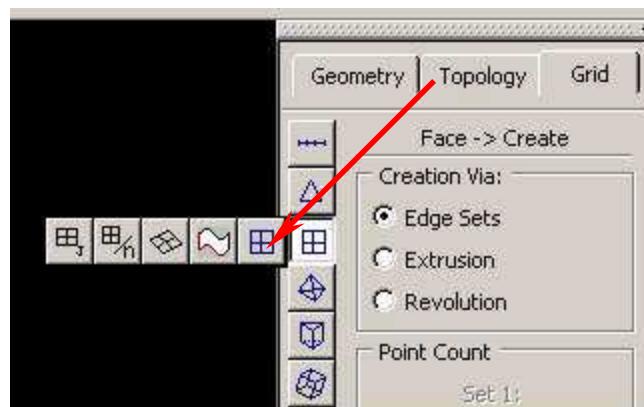


Fig. 7. The view of Create Structured Face panel

Pick the upper wall of the microchannel and apply your choice by using MB, then go against the clock and pick the next wall and repeat the procedure. After the last edge is entered, a cross-hair appears (Fig. 8). The cross-hair is called a handler and indicates that a face has been created.

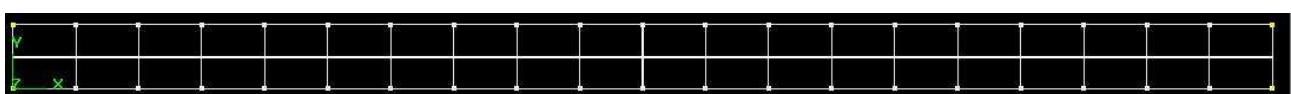


Fig. 8. A cross-hair

The following step is creating a *Structured 2D Block*. Choose that option from *Structured Block Options* (Fig. 9).

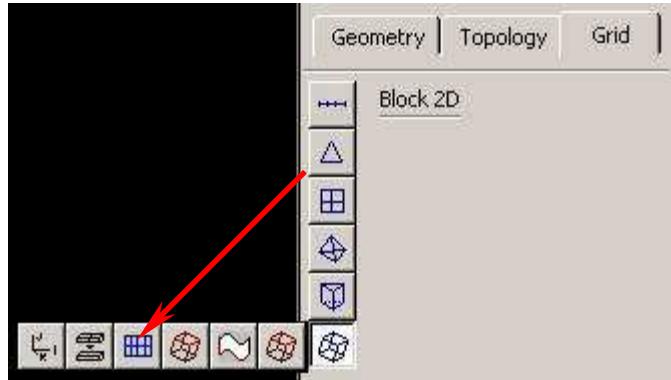


Fig. 9. The view of the Structured Block Options

Pick the cross-hair and accept it by clicking MB, the face turns magenta (Fig. 10), your 2D block is created.

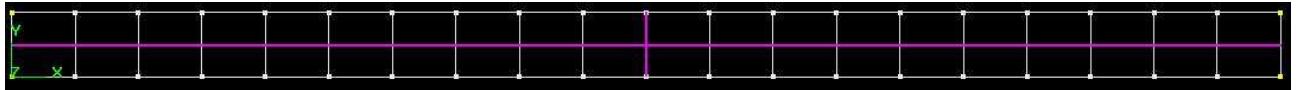


Fig. 10. The view of created 2D block

4. Setting Boundary and Volume Conditions

The last and step in CFD – GEOM is setting boundary conditions. To do this, you should open *Boundary* tab (Fig. 11) and choose *BC/VC Editor* (Fig. 12). Leave the *Volume Conditions* as they are (the inside of the microchannel will be filled in by water solution, so VC should stay as a *Fluid*).

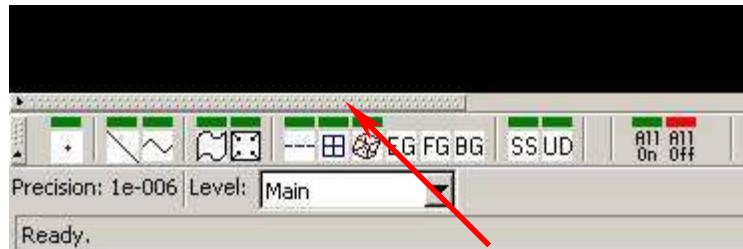


Fig. 11. The view of Boundary tab

In the *BC Editor* change boundaries as it is shown in Fig. 12. Note, that you have to change not only the type of the boundary but also set a new name.

Save the file, save it also as **.DTF**.

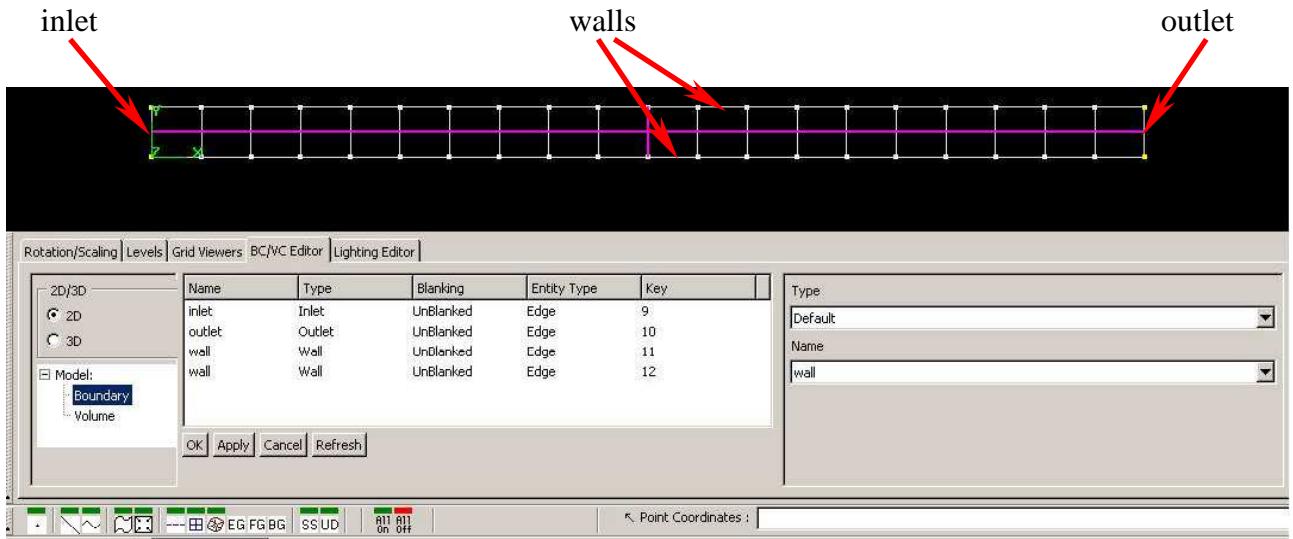


Fig. 12. The BC/VC Editor

You have just created the microchannel filled with the undefined fluid, the channel consists of two walls, inlet, and outlet.

5. Preparing to Simulation of the EOS

Select the CFD – GUI button from the toolbar in CFD – GEOM. This button invokes the CFD – GUI program installed on your machine in CFDRC package. Within CFD – GUI you can apply boundary conditions appropriate for your problem, set up analysis parameters, and start simulation.

In CFD – GUI open your project.

6. Preparing a Mixture

Before you start to set boundary and volume condition, you must define species and mixture, which will be pumped into the microchannel through the inlet.

Click on Tools from the top toolbar and select *Property Manager* (Fig. 13). *Property Manager* opens in a new window, here you define species **SPA-** and mixture **mixA**.

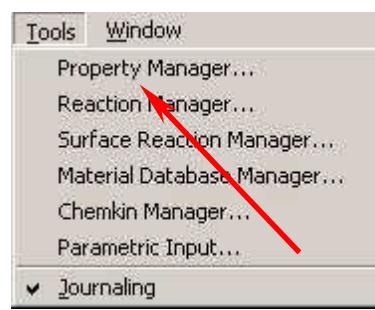


Fig. 13 Tools menu

7. Adding a New Species and New Mixture

In *Property Manager* choose *Species* and click *Add New Species*. Name the species **SPA-** and from default settings change only these shown in Tab. 1.

Tab. 1.

Molecular Weight	1
Mass Diffusivity	1E-11

Click on *Mixtures*, and select *Add New Mixture*. Give a name **mixA** to it. From available species select **SPA-** and press *Add Species* to confirm selection. Set the concentration equal to 1E-6 and confirm creating the mixture mixA by clicking *Apply* button. You will receive a message that concentration does not sum to 1, but you can ignore it, later, in *Volume Conditions*, you define liquid as a water, so your species will be diluted in it. Save all data stored in *Property Manager* in your working directory (e.g. as EOS.PMD).

8. Setting up a Coupled Electrostatic, Flow and Chemical Simulation

As you know, electroosmosis is a phenomena in which electric field affects chemical ions suspended in a mixture.

PT (Problem Type)

Tick modules:

- *Flow*
- *Chemistry*
- *Electric field*

MO (Module Options)

- in the *Shared* panel
 - o change *Transient Condition* from **Steady** into **Transient**
 - o set *Number of Steps* to 100 and *Time Step* to 0.05s
- in *Flow* panel
 - o make sure, that *Reference Pressure* is set to 100000 N/m²
- in *Chemical* panel
 - o change *Chemistry Media* from **Gas Phase** into **Liquid Phase**
 - o make sure, that *Solve Concentration* field is ticked

VC (Volume Condition)

Select the inside of the microchannel:

- in the *Shared* panel
 - o set *Name* to *water*
 - o set *Density Rho* to 1000 kg/m³
 - o set *Viscosity Constant (Kinematic)* to 1.65E-5
 - o set *Relative Permittivity* to 78.5
 - o set *Electrical Conductivity* as *Function of Concentration* to 4E-6 1/Ohm*m
- in *Chemical* panel
 - o change both *Mass Diffusion* and *Mobility* to **constant**

BC (Boundary Condition)

In the graphical window select **inlet**, it turns red:

- in *Flow* panel
 - o change *SubType* from *Fixed Velocity (Cartesian)* to *Fixed Pressure*
 - o leave default parameters (P = 0N/m² and T = 300K)
- in *Chemical* panel
 - o select *Define...* button

- o in *Property Manager* window open your file with previously defined species and mixture and close *Property Manager* window (GUI sometimes loses path to some files, e.g. PMD)
 - o select mixture **mixA** from pull-up menu
- in *Electric panel*
 - o change *Surface Charge* to *Fixed Potential* and set the value of 0V

Select **outlet** from the graphical window:

- leave *Flow* panel as it is (*Fixed Pressure*)
- leave *Chemical* panel as it is
- in *Electric panel*
 - o change *Surface Charge* to *Fixed Potential* and set the value of 50V

Select both **walls** from the graphical window and group them (this allow you to set boundary conditions once to the whole group):

- leave *Flow* panel as it is
- leave *Chemical* panel as it is
- in *Electric panel*
 - o turn on *Electroosmosis*
 - o change *Zeta Potential* to *Electroosmotic Mobility*
 - o set the value of mobility to $-1E-8 \text{ m}^2/\text{V*s}$
 - o leave *Debye Thickness* as it is

SC (Solver Control)

- in *Iteration* panel
 - o set *Max. Iteration* to 10
- in *Solver* panel
 - o change *Species* and *Electric Potential* from **CGS+Pre** to **AMG**
- in *Relaxation* panel
 - o set *Species* to $5E-6$
 - o set *Electric Potential* to $1E-9$
 - o set *Pressure* to 0.9
 - o set *Density* to 0.9
 - o set *Viscosity* to 0.9

Out

- in *Output* panel
 - o set *Timestep Frequency* to 1 (in that case solver will write down every **.DTF** file containing results)
- in *Graphic* panel set the output variables you want to see in CFD – VIEW:
 - o *Velocity Vector*
 - o *Static Pressure*
 - o *Total Pressure*
 - o *Mole Concentration*
 - o All issues in *Electric* frame

Run

Before you run simulation save your file.

Click *Submit to Solver* button to start the simulation.

You can press *View Residuals* or *View Output* buttons to see real-time displays of the residual history and output of the file contents.

9. Simulation Post Processing in CFD – VIEW

Run the CFD – VIEW program and open the DTF in the following way:

- from *File* in top toolbar menu select *Import DTF or PLOT3D*
- in *Open Data File* select *name.0000.DTF* file (where *name* is the name of your file)

Before starting the visualization, disable *Fit on Read* option from *Miscellaneous* menu (Fig. 14). This option causes coming back to default settings of the CFD – VIEW that makes visualization process quite troublesome.

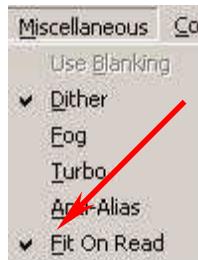


Fig. 14. Fit on Read option

In the *Objects* window in bottom-right corner, select *Geom*.

From the toolbar select following buttons:

- *Flooded on*
- *Contour on*

Select the variable you want to display in Primary var. window – in this simulation – choose **SPA-mole**.

You may increase the number of contours for better viewing of distribution of the **SPA-** sample:

- from the Control Panel located in the bottom, select Colormap (sixth from the left)
- increase the number of contours from 10 to 100

You can zoom in or out by using MB.

Your simulation is a transient one, so you can animate results. Select *Animate Data Files...* from *Gizmos* menu (Fig 15). The *Animate Data Files* window appears (Fig. 16), set *Data Files Range* from 0 to 50; *Current* 0; *Increment* 1. To see animation select *Play* button.

Part Two

EPH Simulation

10. Generating the Microchannel

The microchannel used in this part is a little bit different from the previous one. It is divided into two parts: first (a small one) is a reservoir for a mixture, second one is a capillary in which EPH phenomenon will be observed.

In CFD – GEOM draw six points: (0; 0), (0; 5e-5), (-5e-5; 0), (-5e-5; 5e-5), (2e-3; 0), (2e-3; 5e-5) and then connect them with the polyline. Results of your work should be similar to the one presented in Fig. 15

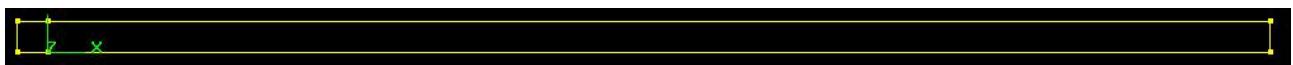


Fig. 15. A structure generated in CFD – GEOM

To each wall of the reservoir set 2 grid points, and to two longest walls of the capillary set 41 grid points. Continue the mesh generation of the device like in the Steps 3 and 4 of this tutorial. In VC/BC Editor change names of volumes into reservoir and capillary, for boundaries set names like in Fig. 16.

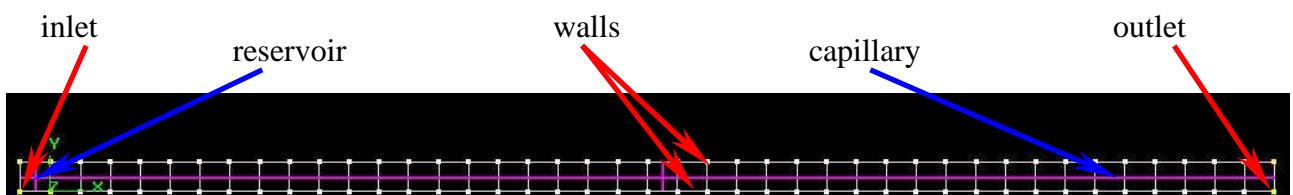


Fig. 16. The view of the volumes and boundaries of the device

11. Preparing to the Simulation of the EPH

Select the CFD – GUI button from the toolbar in CFD – GEOM. This button invokes the CFD – GUI program installed on your machine in CFDRC package. Within CFD – GUI you can apply boundary conditions appropriate for your problem, set up analysis parameters, and start simulation.

In CFD – GUI open your project.

12. Preparing a Mixture

Before you start to set boundary and volume condition, you must define species and mixture, (just like in Step 6)

Click on Tools from the top toolbar and select *Property Manager* (Fig. 13). *Property Manager* opens in a new window, here you define species **SPB-**, **SPC-**, **SPD-** and mixture **mixB**.

13. Adding a New Species and New Mixture

In *Property Manager* define species presented in Tab. 2.

Tab. 2

SPB-	Molecular Weight	1
	Charge Exchange Cross Section	40
	Mass Diffusivity	1E-11
	Mobility	1E-8
SPC-	Molecular Weight	1
	Charge Exchange Cross Section	40
	Mass Diffusivity	1E-11
	Mobility	2E-8
SPD-	Molecular Weight	1
	Charge Exchange Cross Section	40
	Mass Diffusivity	1E-11
	Mobility	4E-8

Define mixture **mixB**, which contains species **SPB-**, **SPC-**, **SPD-**. To each of these species set concentration equal to 1E-6.

Save all data in your working directory (e.g. as *EPH.PMD*).

14. Setting up a Coupled Electrostatic, Flow and Chemical Simulation

As you know, electrophoresis is a phenomena in which electric field affects chemical ions suspended in a mixture.

The EPH simulation in CFD – GUI is very similar to the EOS simulation, so in this Step, only differences in setting parameters described previously in Step 8 will be presented.

MO (Module Options)

- in the *Shared* panel
 - o change *Transient Condition* from **Steady** into **Transient**
 - o set *Number of Steps* to 100 and *Time Step* to 0.05s

BC (Boundary Conditions)

In the graphical window select **inlet**, it turns red:

- in *Flow* panel
 - o make sure, that in *SubType Fixed Velocity (Cartesian)* is chosen
- leave *Chemical* panel as it is
- in *Electric* panel
 - o change *Surface Charge* to *Fixed Potential* and set the value of 0V

Select **outlet** from the graphical window:

- leave *Flow* panel as it is (*Fixed Pressure*)
- leave *Chemical* panel as it is
- in *Electric* panel
 - o change *Surface Charge* to *Fixed Potential* and set the value of 100V

Because you want to simulate EPH phenomenon, **do not** turn on *Electroosmosis* option, when grouping walls.

IC (Initial Conditions)

- change **IC Global Settings** from *For All Volumes* into *Volume by Volume*
- select **reservoir** volume
 - o in *Chemical* panel select mixture **mixB**

After setting all items, save the file and run simulation.

Run CFD – VIEW to visualize results.

Part Three

Projects

15. Project 1

The aim of this project is to use basic knowledge of designing and simulating microchannel structures in CFD software to design a simple cross-like biochip. The blueprint of microchannel is shown in Fig. 17.

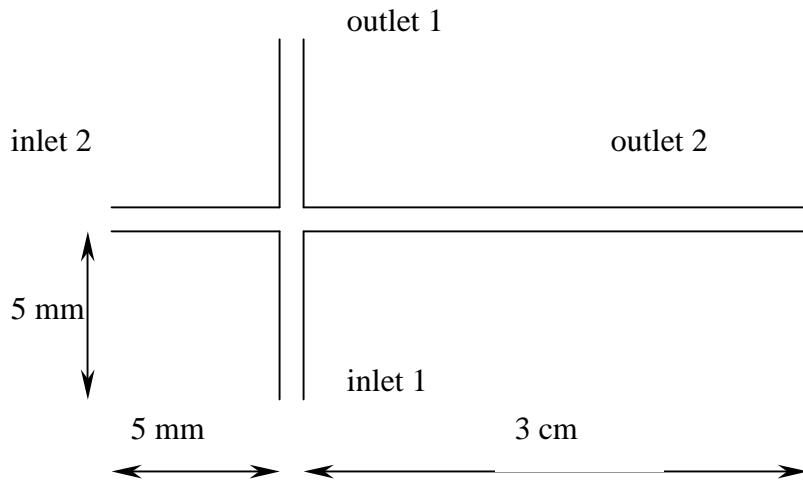


Fig. 17. The blueprint of the project

The width of microchannel is 50um, the mesh should be structured.

In the first phase, the vertical channel should be filled up with the mixture in electroosmotic way, then, after changing boundary conditions, the sample should be sucked into the horizontal channel. In this long channel separation process should be visible.

Tips:

- In the first phase **inlet2** and **outlet2** should be defined as walls; to **inlet1** and **outlet1** potentials of value $V_{inlet1} = 0V$, $V_{outlet1} = 500V$ should be set. In the second phase **inlet1** and **outlet1** should be defined as walls and to **inlet2** and **outlet2** ($V_{inlet2} = 0V$, $V_{outlet2} = 1000V$).
- After first simulation remember to:
 - o turn off *Electroosmosis* in **BC**
 - o change **IC (Initial Conditions)** from *For All Volumes* to *Volume by Volume*
 - o change *Mobility* in species

16. Project 2 – for Students Scientific Association

Prepare and run multi-capillary biochip simulation in which a comparison among different species in different capillaries will be possible.