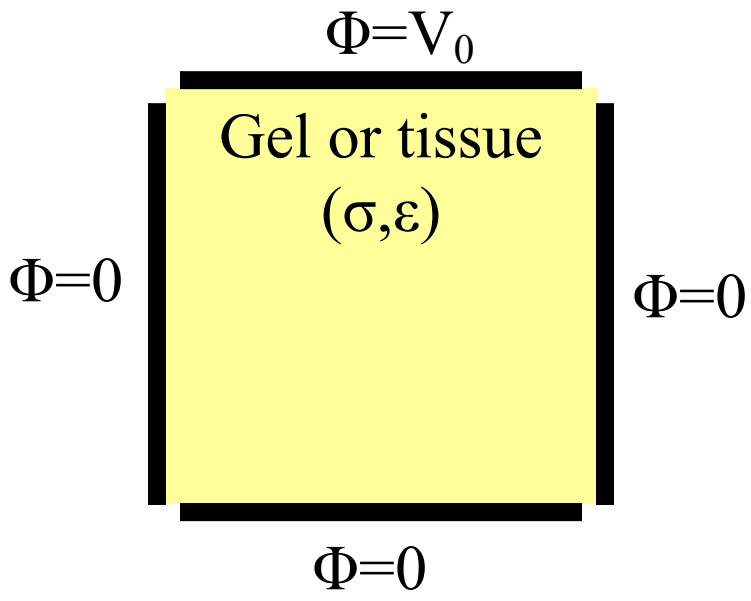
We've learned how to solve the problem below by using separation of variables. Now we can solve the same problem using the finite element model in FEMLAB.



$$\nabla^2 \Phi = 0$$

$$\vec{J}_e = -\sigma \nabla \Phi$$

For the analytical solution, please see lecture notes.

Click on **COMSOL Multiphysics 3.3** on your desktop

In **Model Navigator**, under **New**,

- choose either 2D or 3D space dimension
- under **Electromagnetics**, choose either **Electrostatics** or **Conductive Media DC**

Draw->**Specify Objects**->**Square**-> specify the size and position of the square you want to draw

(If you want to create a composite object , i.e. a square + a circle overlapping: go to **Draw**->**create composite object**, then select all the objects you want to be in the composite (by holding Ctrl), and click on **Union**, also uncheck **Keep interior boundaries**, then click **OK**.)

Physics->**Subdomain Settings**:

-Select **Subdomains** (since you only have a square in this case, it's the subdomain "1")
-Click on "**σ (isotropic)**", then enter a value for electrical conductivity in the **Value/Expression** box.
-Click OK.

Physics->**Boundary Settings**:

-For each boundary (i.e. 1, 2, 3, 4), select the appropriate **Boundary condition** (i.e. current flow, inward current flow, distributed resistance, electric insulation, electric potential, ground).
-also fill in **Value/Expression** if applicable.
-Click OK.

Mesh->**Initialize Mesh**

Solve->**Solve Problem**

Postprocessing->**Plot Parameters**

-**Surface**: check **Surface plot**; at **Predefined quantities**, choose **Electric potential**.
-**Streamline**: check **Streamline plot**; at **Predefined quantities**, choose **Electric field**; you can also change the number of streamlines by specifying the **Number of start points**
-**Arrow**: check **Arrow plot**; at **Predefined quantities**, choose **Electric field**; you can make the arrows bigger or smaller by unchecking **Auto** (under **Arrow parameters**) and enter a **scale factor**.
-Click OK

You should get plots similar to the ones shown here.