MNIS – Introduction to finite element modeling

Parallel plate capacitor

What are we going to learn:

- -get familiar with Comsol interface
- -how to define material properties and boundary conditions
- -use of parameters and parametric sweeps
- -calculating electrical capacitance

Starting from version 4.4 the Comsol interface looks something like this:

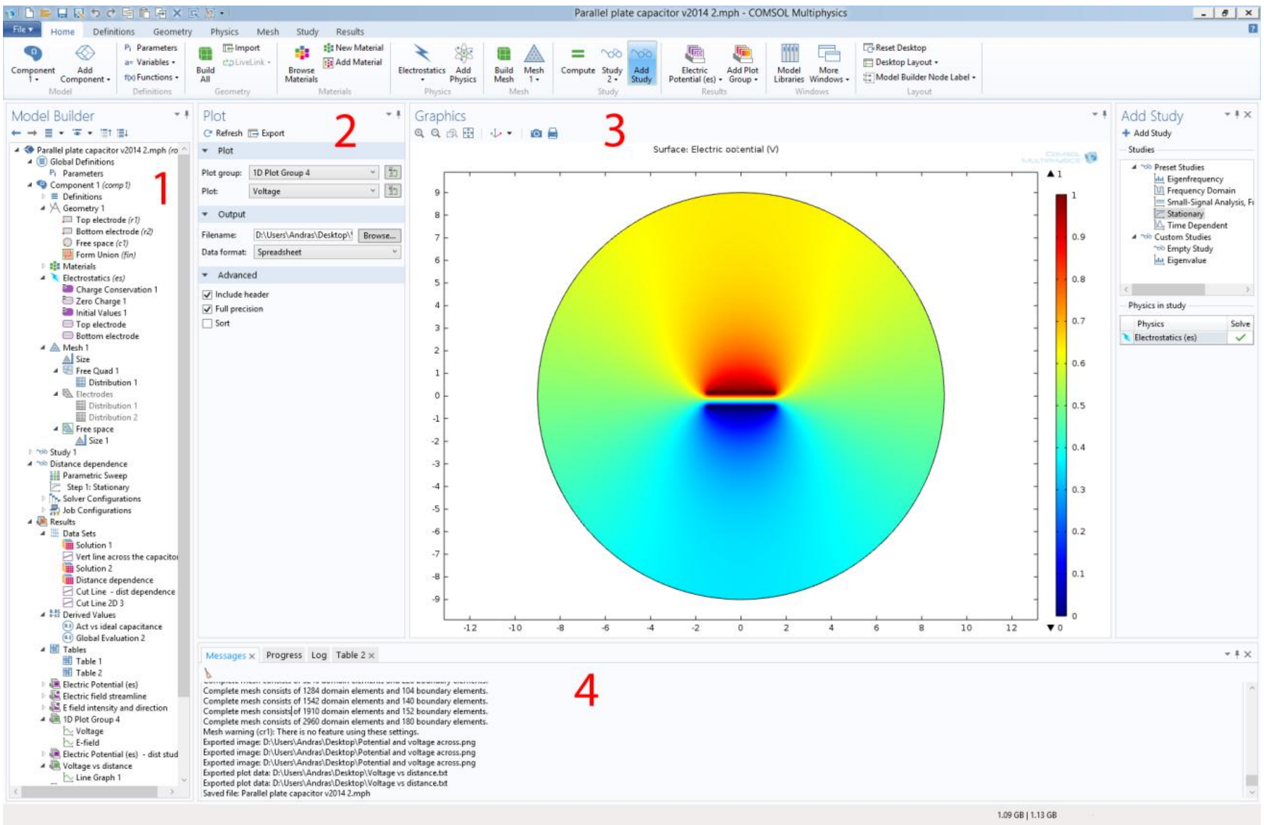


Fig. 1. Comsol interface

and is divided in several panes:

1: **the model builder** – contains all the definitions related to your model organized by default in a tree-like view
2: the **settings** pane – when you choose an item in model builder, this is where you can change the related parameters
3: **Graphics** pane – this is where your model or the graphical representation of the results are shown
4: Message window, progress, results – messages and numerical results of calculations are shown here

Let us now build a simple model with a parallel plate capacitor. This is a simple example but because we are already very familiar with the expected results, it is a good starting point for learning how to use Comsol and perform finite element modeling.

1. BUILDING THE MODEL

First, we need to define the type of geometry (2D vs 3D) and the type of physics that we are going to use in our model (for example heat transfer, mechanics or electrostatics). To do this, start Comsol and start the model wizard. Choose **2D** under space dimension, **AC/DC > Electric Fields and Currents ► electrostatics (es)** under **Select physics**. Press **add**, and then click on **study.** Select **General studies ► Stationary** and click on **Done**.

We should now build the model. It will consist of two parallel plates and an area within which we will perform our calculations. Let us define the first plate. Right click on **Geometry 1** (or similar) in the **model builder**. Choose **rectangle.** In the settings, insert **3** m for **width** and **0.1** m for **height**. Choose **Center** under **Position ► base**. This will make it easier for us to center everything. Enter **0.5** m for **y**. Build the rectangle by clicking on the **build selected** or **build all objects** icon on top of the **settings pane.** Rename **rectangle 1** by pressing F2 and enter **Top electrode** as a new name. As your model becomes more and more complex, it will be difficult to keep track of things. It is therefore useful to develop a habit of giving more meaningful names to different parts of your model.

You should now add a second rectangle in a similar way, with the same dimensions but positioned at 0, -0.5 . Rename this rectangle **Bottom electrode.**

This would define the capacitor. However, in order to perform modeling, you also need to define a finite area that would surround your capacitor. All the calculations (for example potential calculations) would then be performed in this area. To do this we can choose any shape, a circle will do well, so add a circle with radius **5** m, centered at 0,0 to the geometry.

You should now have a geometry that looks like this:

Fig. 2. Parallel plate capacitor model surrounded by a circular area within which we will perform the calculations.

We can now proceed to the next step. First, we must define material properties.

2. DEFINING MATERIAL PROPERTIES

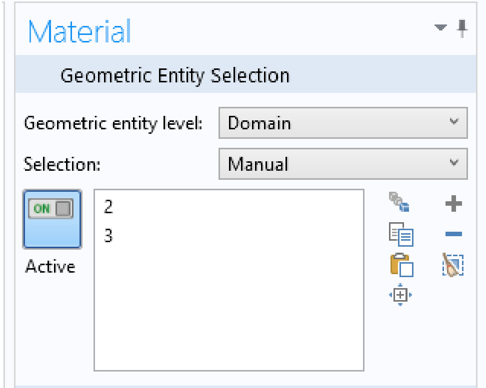
Both plates will be made of a conductive material while the surrounding area will be made of air. For a conductor, any metal such as for example copper would do.

We will first load material definitions into the model by right-clicking on **Materials** in the **Model builder** and selecting **Add Material from Library**. Enter **copper** in the search box and choose **copper** either from the **built-in** section or the **AC/DC** section. Select the two electrodes and click on **add to selection**.

Repeat the process by selecting the circle and assigning **air** to it.

You can revisit your material assignments by clicking on the corresponding materials in the **materials section** of the model builder.

To assign copper to a plate, select it first by left-clicking on it in the **graphics** window. The object should turn pink. You can now assign the material by doing a right-click anywhere in the **graphics** window. The selection window should now contain the two plates (2 and 3) and look something like this:



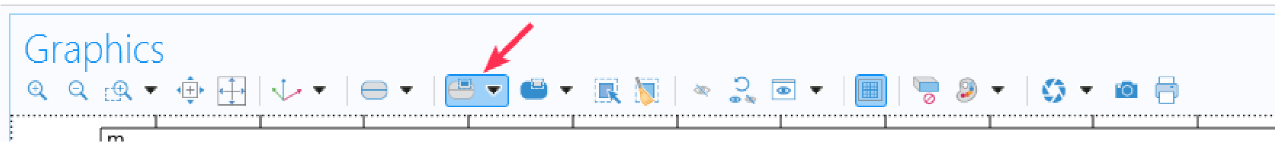
If you made a mistake, you can remove the material assignment by choosing the corresponding object in the **selection** window and clicking on the minus sign. To re-assign copper to an object, just click on it.

We could also define our own materials with appropriate dielectric functions and conductivities, using the **New material** option in the menu that appears when you right-click on **materials** in the **model builder**.

We can now move to the next step and define the boundary conditions.

3. DEFINING BOUNDARY CONDITIONS

Both plates will have fixed, defined voltages. To define this, you should right-click on **electrostatics** in the **model builder** and select **terminal** boundary condition (near the middle of the pop-up menu). You could also in principle use the **electric potential** condition, except that with terminal it is a little easier to calculate the capacitance. Assign all four edges of the top electrode to **boundary selection.** You can select them using the select box tool from the **graphics** window.



Choose **voltage** under **terminal type** and leave $\mathbf{V_0}$ at 1V. Rename **Terminal 1** to **Top electrode voltage**. Create another **terminal** boundary condition and assign the bottom electrode to it. Rename it to **bottom electrode voltage**. Define the potential as **0** V. You should now be in a position to calculate the potential distribution by right-clicking on **study 1** (or similar) in the **model builder** and selecting **compute** or clicking on the **compute** icon in the toolbar, **Home>study** section**.** You should get something like this in the **graphics** window:

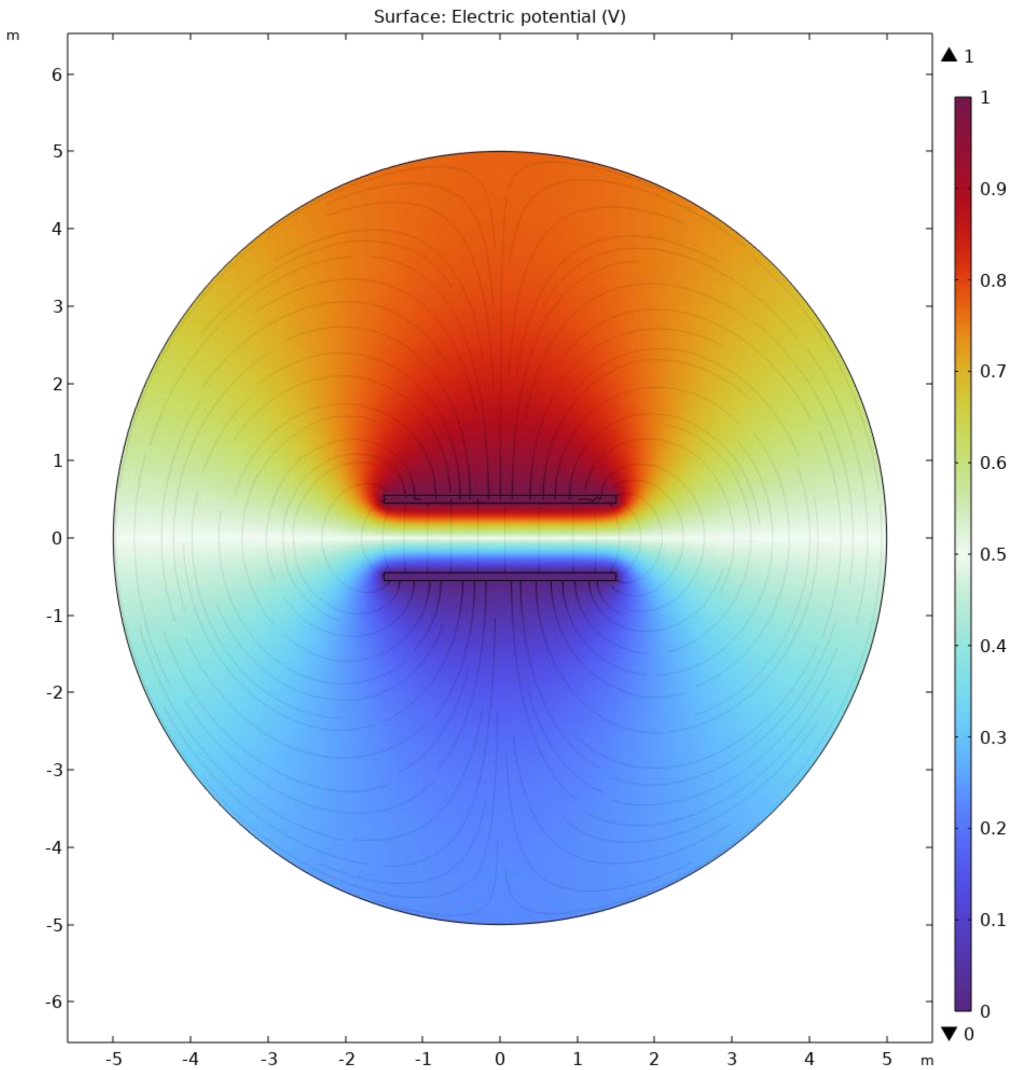


Fig. 3. Potential distribution in the space surrounding the parallel plate capacitor.

In case you get an error message, you most probably forgot to assign a material to one or more of the regions in your model.

We will now explore different ways of representing the result of FEM calculation, but before that let us look at the mesh that was used for the calculation.

4. MESH FOR FINITE ELEMENT MODELING

Comsol has built-in algorithms that take care of the mesh geometry and in most of the cases you do not need to worry about it. You can access the options related to the mesh by clicking on **mesh 1** (or similar) in the **model builder**. In our case the mesh looks something like this:

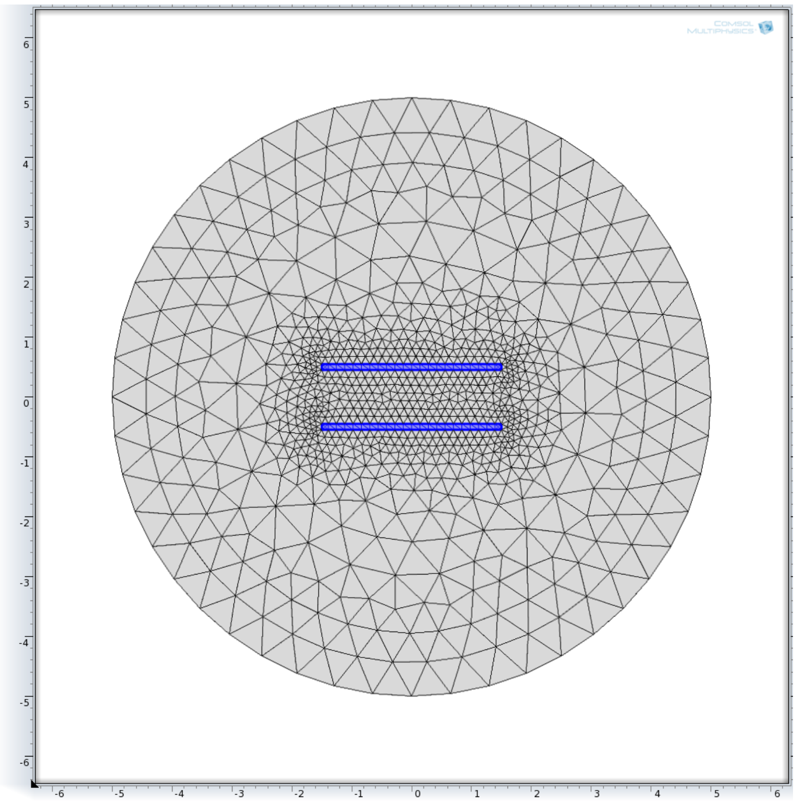


Fig. 4. Mesh used for calculations.

In case you are not happy with Comsol's mesh, you have several options. You can modify the mesh density by selecting **physics-controlled mesh** under **Mesh** and for example selecting **extra fine** under element size. Do not forget to click **build all** to update the mesh.

You can also select **user-controlled mesh** under **sequence type** in **mesh settings** for more control. To change the mesh parameters here click on **model builder ► mesh 1 ► size** and choose **custom** under **element size.** This will give you access to various parameters under **element size parameters.**

We will also briefly review some advanced ways of customizing the mesh. In many of the projects that you will study, you will encounter very thin objects that you will have to mesh differently from the free space. Let us customize the meshing for and around the electrodes independently from the rest of the system. First, let us delete the existing mesh by righ-clicking on **Mesh 1** in the model builder and selecting **delete**. We should now add a new mesh by right-clicking on **Component 1** in the model tree and selecting **Add mesh**. Right-click on **Mesh 1** and select **Free triangular**. In the **Free Triangular** pane, choose **Domain** under **geometric entity level** and select the two electrodes. Right-click on **Free triangular 1** and select **size**. Under **size**, you can now adjust the size of this mesh, for example to **extra fine**. Let's also rename the **Free Triangular 1** mesh to **Electrodes**. Repeat this for the circle and set the size to extremely coarse.

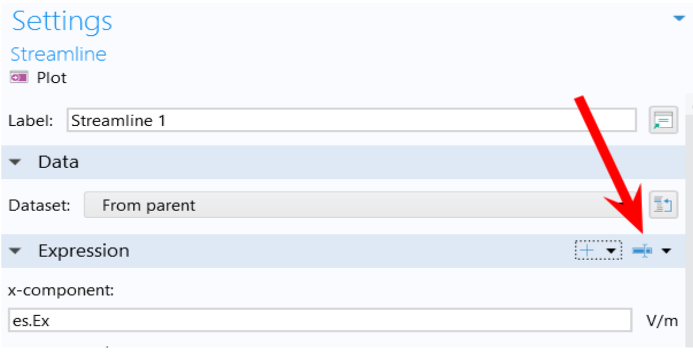
Another potentially useful way of meshing is to define the number of elements over an interesting edge in our model, for example the sides of the electrodes, where we could have a higher density, for example 10 triangles along this line. To do this, right click on the **electrodes** mesh, select **distribution** and select the left and right edges of the two electrodes.

Sometimes you can also use your knowledge of physics to reduce the mesh complexity and adapt it to the material and the geometry. For example, we know that inside the metal electrodes there will be no electric field and the potential will be constant. In addition, the rectangular geometry is actually more suited to a different type of mesh, with Quad mesh elements instead of triangles. To assign this type of a mesh to the electrodes, disable the electrodes mesh by right-clicking on disable in the pop-up menu. Right click on **Mesh 1,** select **Free quad** and asign it to the electrodes. Rename **Free quad 1** to **Electrodes quad.** Create a **distribution** condition under electrodes quad and assign it to the top and bottom edges of the two electrodes. Put a reasonable number, for example **20** under the **number of elements**. In case the mesh is not being properly updated, you should also make sure that the **electrodes quad** is on a higher position than **Free space** because this way, it will be the distribution in the electrodes that will be driving the meshing in the free-space region near the electrodes and not the other way around.

Let us go back to different ways of representing the results from modeling.

5. PLOTTING THE RESULTS

After having performed the calculations of the potential, Comsol stores the data under **Results ► Data sets ► Solution 1** in the **model builder.** This solution contains a matrix of potential values for the entire model. From these values, you can calculate other quantities such as for example the electric field, energy stored in the field, capacitance etc. Sometimes you would also want to plot the values of the potential along some characteristic direction or in a given point. To do this, you need to define a data subset of the solution which would correspond to the line or point and then create a line graph along the line. Let us first review 2D plots because for these we do not need to define any data subsets. Here is how you would create a new plot of the electric field lines. Right click on **Results** and choose **2D plot group.** Rename it **streamline.** Right click on **2D Plot group 2** that you just created and select **streamline.** The software is intelligent enough to know that most people would use this kind of a plot to visualize the electric field, so the **x component** and **y component** under **expression** already contain the right expressions (**es.Ex** and **es.Ey**). You also have the option to select other expressions from the menu that can be accessed by clicking on this icon



and selecting a set of expressions such as for example **electrostatics ► electric ► electric displacement field**. We now need to define sources and sinks for the field lines. For this, you should select all the boundaries corresponding to the parallel plates in the **graphics** window. The **selection** section should now be populated with boundaries that form the two parallel electrodes. You can also increase **number** under **streamline positioning** to something like 40 to have more streamlines. You can now update the plot by pressing on the plot icon under **streamline** settings window title. Your result should look something like this:

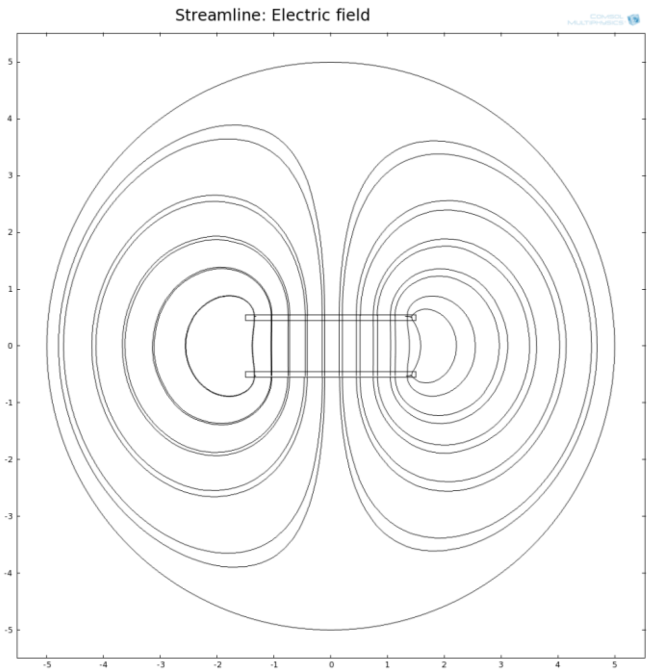


Fig. 5. Plot of the electric field streamlines.

Another interesting way of plotting the data would be to create a color plot with electric field intensity and use arrow lines to show their direction. For this let's create another **2D plot group** and add a **surface** to the plot group. Select **electrostatics ► electric ► electric field norm** by clicking on the replace expression icon and then create the plot by pressing F8. Let's now add an **arrow surface** plot in the same group. The default setting results in a rather boring graph, so let's increase the number of **points** under **arrow positioning ► x grid points** and **y grid points** to something like 50. Here is the resulting plot:

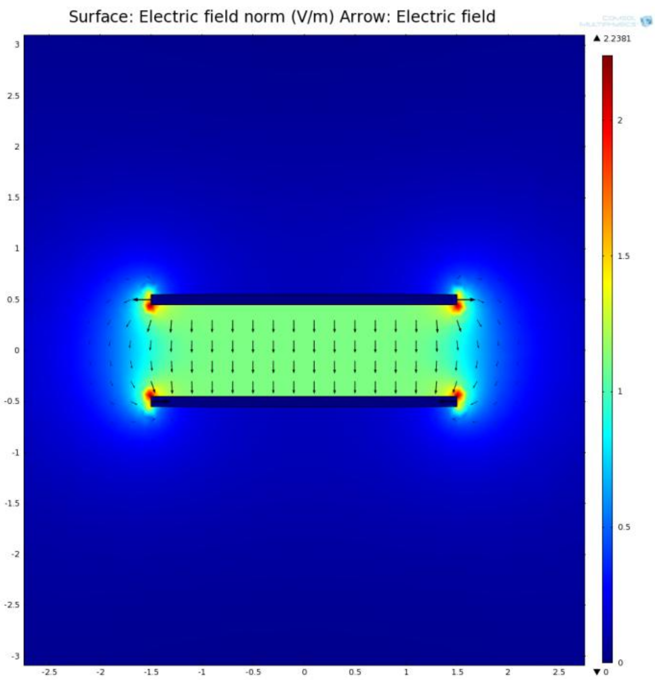


Fig. 6. Electric field vectors and intensity

In order to improve our bookkeeping, we can also change the names of these plot groups that we have just created by right clicking on them and selecting **rename** to something like **E-field lines** and **E-field intensity and vectors**, so that we can get an idea of what is in the plot.

Finally, let us create plots of magnitude of the potential and the electric field along a line that goes across the capacitor. First, we need to create a subset of the data by right-clicking on **Results ► data sets** and selecting **cut line 2D.** Under **line data** enter **0,-5** for point 1 and **0, 5** for point 2. You now need to create a **1D plot group** by right clicking on **results** in the **model builder.** Under **data ► data set** you need to select **cut line 2D 1,** the cut line that you just created. You now need to create a **line graph** under the **1D plot group.** Under expression, you can leave the default expression (**V**). For the x-axis data, choose **Expression** under **Parameter** and enter **y** so that the potential is plotted as a function of the y coordinate. Plot the graph. You can now add a plot for the electric field in the same plot group or create a separate 1D plot group only for the electric field. In the **expression** under **y-axis data** you need to choose **electric field norm** or type in **es.normE**.

The result should look something like this:

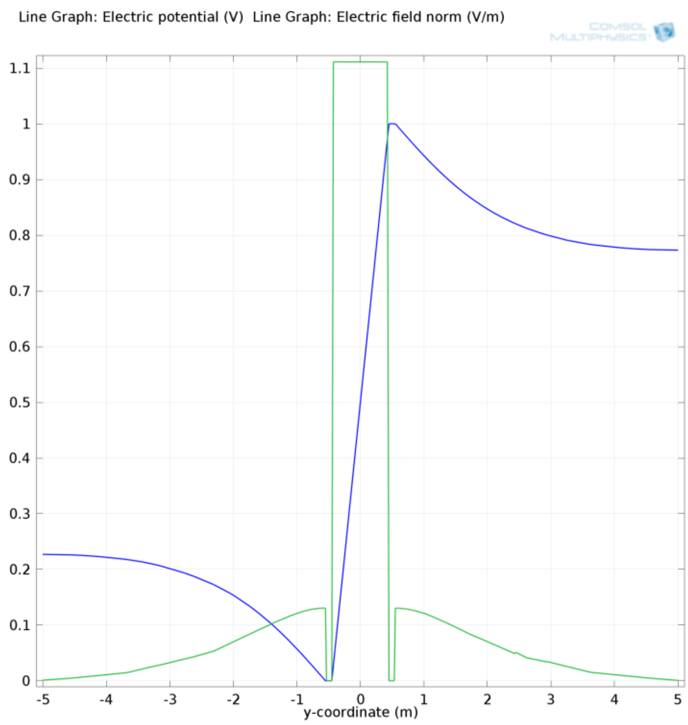


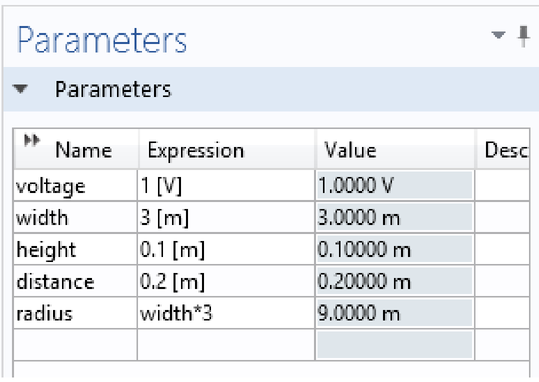
Fig. 7. Electric field (green) and the potential (blue) along a line crossing the parallel plate capacitor.

Finally, let us rename the plot groups and plot, so we can keep track of things more easily. I have renamed my group to **Field and potential across** and the plots to simply **V** and **E norm.**

6. PARAMETER-DRIVEN MODELING

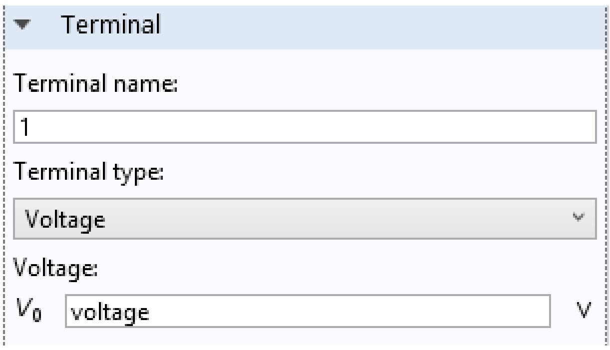
This is all nice, but if we wanted to change something on the model like the potential between the electrodes or the separation between the plates, we would have to dig through sections of the **model builder** and change these parameters manually. There is of course a more intelligent way of doing this. We can define a set of parameters, define their values in one place and replace all the numeric values we had to type in with parameter names. This gives us a way to easily change something on the model and also to perform more sophisticated studies; for example, we could look into how the capacitance changes with plate separation.

Parameters can be defined by right clicking on **model builder ▸ global definitions** and selecting **parameters.** In the table **parameters** we can enter the names, values and descriptions for different parameters – let us define them for voltage and the geometry. Let us enter **voltage** under name, **1 [V]** under expression. Value should now display **1 V.** Attention: everything is case sensitive. In case your unit (for example [v]) is not recognized by Comsol it will be colored orange. We should now add parameters **width** (expression **3 [m]**), **height 0.1[m]**, **distance 0.2 [m]**, **radius (expression width*3).** The result should look like this:

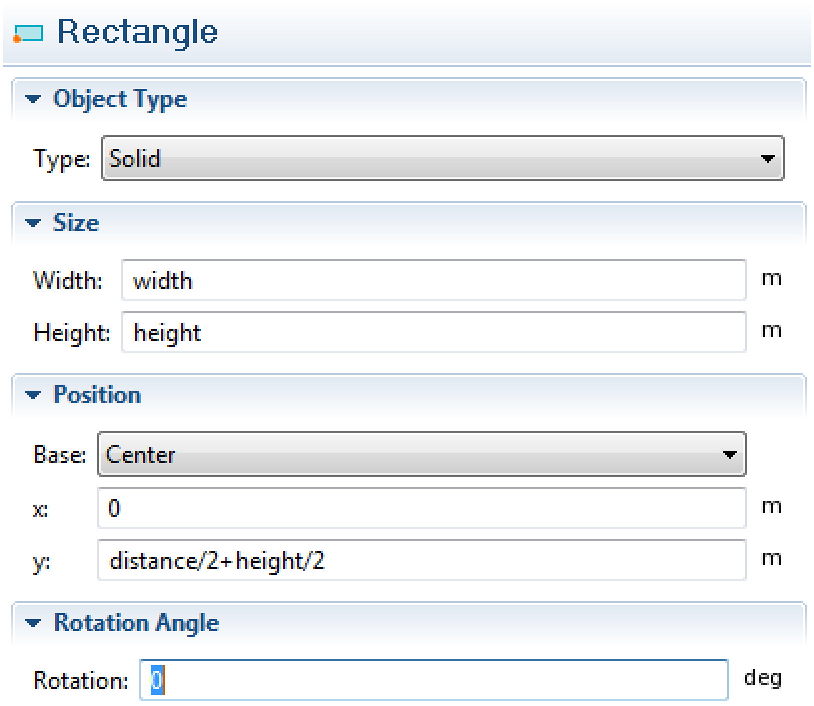


Name	Expression	Value	Desc
voltage	1 [V]	1.0000 V	
width	3 [m]	3.0000 m	
height	0.1 [m]	0.10000 m	
distance	0.2 [m]	0.20000 m	
radius	width*3	9.0000 m	

We now need to use these parameters in the model. Let's go back to where we defined the potential on the top plate to be 1V (**electrostatics ▸ terminal 1 ▸ terminal ▸ V0**) and replace 1 with **voltage**:



Let's also replace the geometric parameters: put **width** instead of 3 under **geometry1 ▸ top electrode (or rectangle1)** instead of 3, same for **height.** For **position ▸ y** enter **distance/2+height/2** for the top plate and **-(distance/2+height/2)** for the bottom plate:



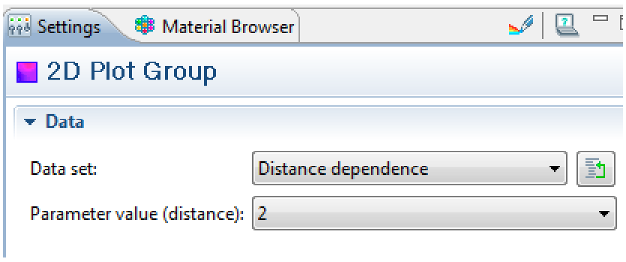
Finally, you should enter **radius** for the circle1 radius and press **build all objects.**

You can now modify these parameters easily in the table under **global definitions ► parameters.** A much more important application is in parametric sweeps, where you can model how the capacitance of the capacitor changes with the geometry.

To do this, let us define a new study by right-clicking on the top section in the **model builder** that carries the file name. A small panel will open on the right hand side of the Comsol window. Select **stationary** under study type and **add study**. Right click on **study 2** and add **parametric sweep.** Add **distance** under parameter names by clicking on the plus sign. Enter parameter values by clicking on this icon next to **parameter values.** Enter **0.1** for **start, 0.1** for step and **2** for **stop**.

Click **replace**. Let us also rename study 2 **distance dependence.** Start the calculation by right-clicking **compute** under **distance dependence.**

After the calculation has ended, you should see new solutions appearing under **results ► data sets**. The one with the highest number contains your distance parametric sweep. Let's rename it **distance dependence.** To plot the new results (without losing the old ones) we need to create new plot groups. Alternatively, you could also go back to your old plots and choose **distance dependence** under for example **2d plot group ▸ data ▸ data set.**



Let us create a plot of voltage across the capacitor vs. the distance. First, create a new cut line data set associated with the **distance dependence** data set, in the similar way you did when plotting e-field and voltage across the capacitor: right click on **data sets** and select **cut line 2D**. Enter 0,-radius and 0,radius as point coordinates and make sure the data set is **distance dependence.** Now create a new 1d plot group, choose **cut line 2D 2** as data set and create a new **line graph** in this plot group. After you click on the plot icon you should get something like this:

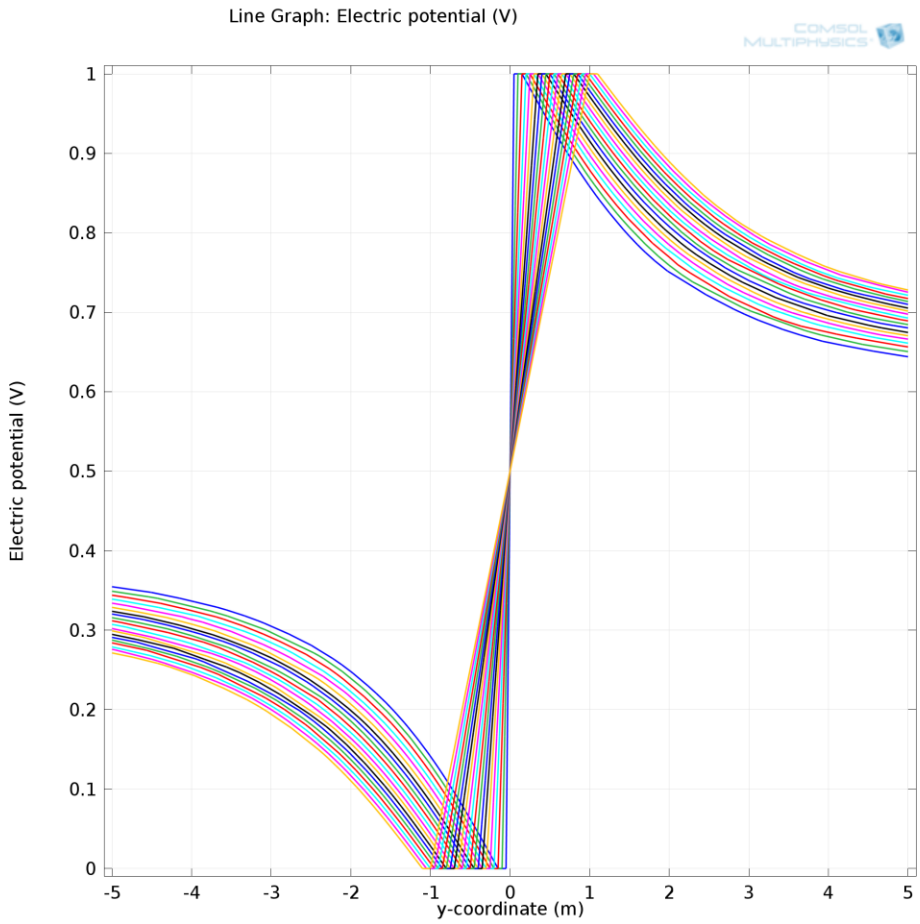


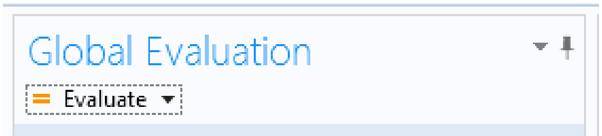
Fig. 8. Electrical potential across the parallel plates vs. the distance

We can also calculate the capacitance. This kind of calculations that involve integration or average over a large area in the model can be set up under **results ► derived values.** Let's right click on **derived values** and add a **global evaluation.** Choose **distance dependence** under **data set.** Under **expression,** you can enter **electrostatics ► terminals ► capacitance (es.C11).** In case you decided to use the **electric potential** boundary condition or if you have another version of Comsol, this expression might not be available or recognized. In this case, you can evaluate capacitance through electrostatic energy via:

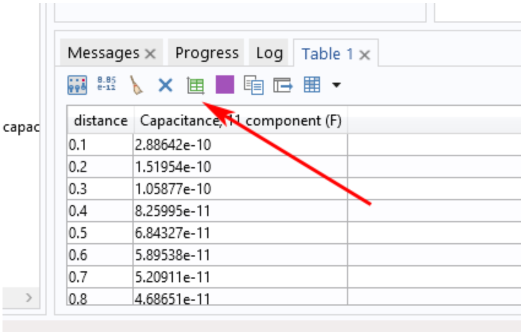
$$E = \frac{1}{2} C U^2$$

By entering **2*es.intWe/voltage^2** in the expression (total electrostatic energy es.intWe is available from the **replace expression** menu.

After pressing evaluate



you should see a table with different capacitance values displayed in the lower right corner of the comsol window. You can plot their dependence on the distance by clocking on the plot icon:



distance	Capacitance, 11 component (F)
0.1	2.88642e-10
0.2	1.51954e-10
0.3	1.05877e-10
0.4	8.25995e-11
0.5	6.84327e-11
0.6	5.89538e-11
0.7	5.20911e-11
0.8	4.68651e-11

It should result in this:

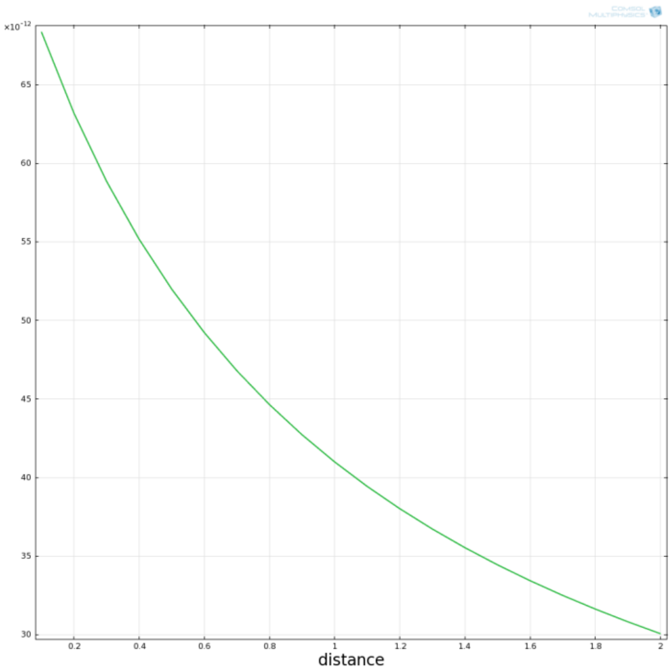


Fig. 9. Capacitance as a function of distance

We can now also compare the calculated capacitance values with those from the widely used formula:

$$C = \frac{S\varepsilon_0}{d}$$

For this, we can define a new **global evaluation** with the following expression:
es.C11*distance/(width*8.85e-12). This will be the ratio of modeled, actual capacitance to the ideal one from the parallel plate model.

In our case the result will look something like this:

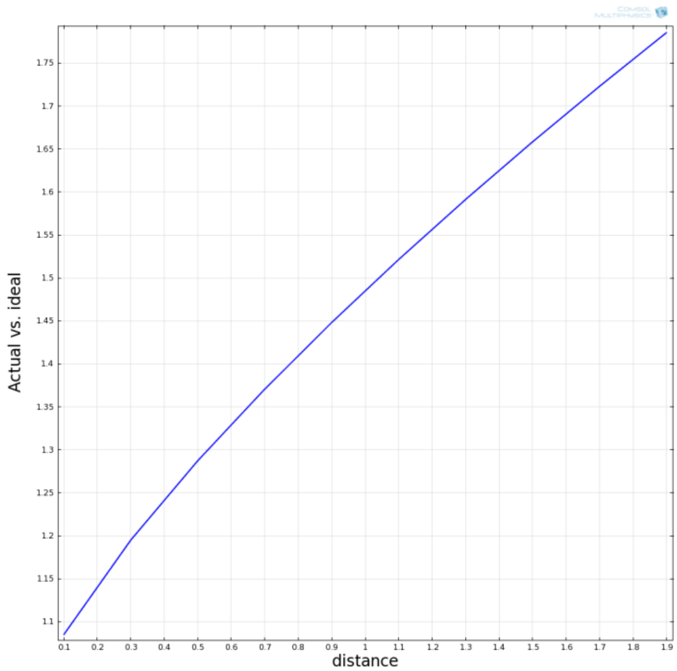


Fig. 10. Actual vs. idealized capacitance.