CAE of Sensors and Actuators
Date: November 2019

Lehrstuhl für Sensorik
Universität Erlangen-Nürnberg
Daniel Kiefer

Comsol Tutorial: Electrostatic Field

In this example, we would like to simulate the electric field and the capacitance of a **cylindrical** capacitor, including the surrounding air. The inner electrode consists of a centered rod and the outer electrode is the metallic coat. Between the inner and outer electrodes a voltage of 1 V shall be applied.

The dimensions are as follows

- ○ radius of inner electrode `r_inner` = 1 mm
- ○ outer radius of outer electrode `r_outer` = 6 mm
- ○ thickness of outer electrode `h_outer` = 1 mm
- ○ height of cylinder `height` = 100 mm
- ○ size of additional boundary `h_boundary` = 10 mm
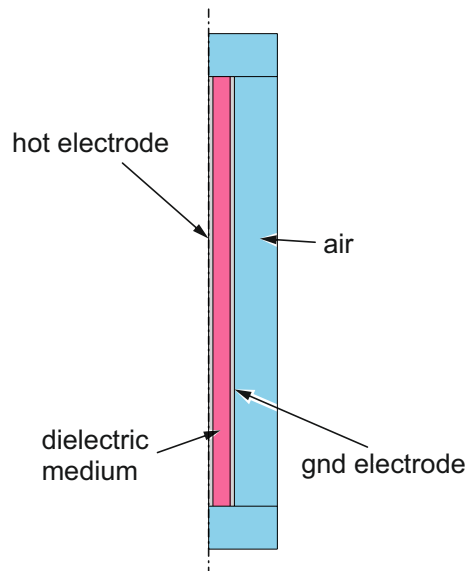

Figure 1: Domain regions of the capacitor model

This tutorial is divided into four parts:

- Part I. Modeling the Geometry and Meshing

- Part II. Simulation

- Part III. Postprocessing and Visualization

- Part IV. Validation

Further information: Refer to the following PDFs linked in StudOn

- *Introduction to COMSOL Multiphysics*

- *COMSOL Multiphysics Reference Manual*

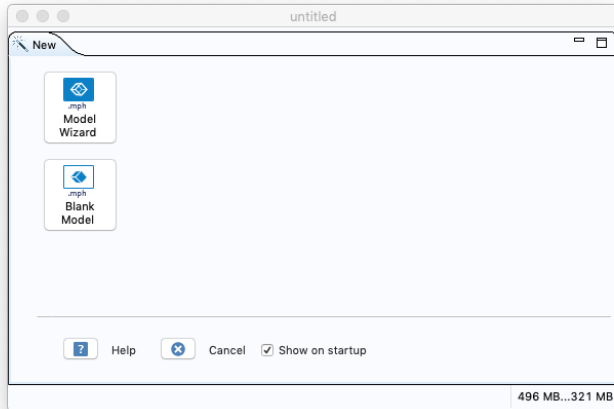# 1 Part I. Modeling the Geometry and Meshing

Setting up the model

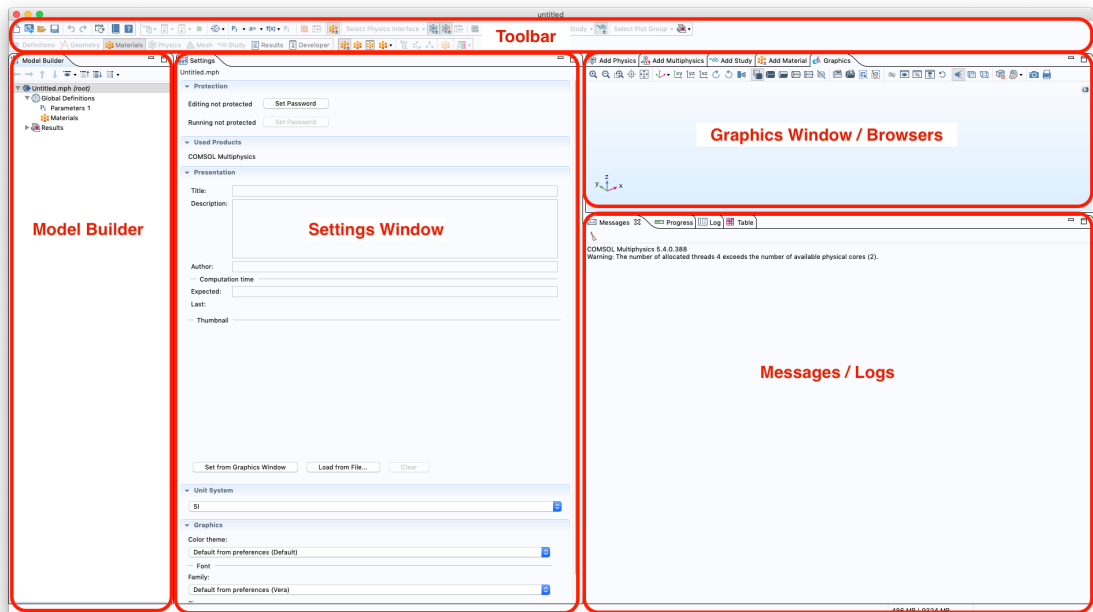We create a new COMSOL model and set up required parameters and components.

1.1. open COMSOL Multiphysics

- if you can't find it in the start menu: open a terminal, type `comsol` and press the enter key.
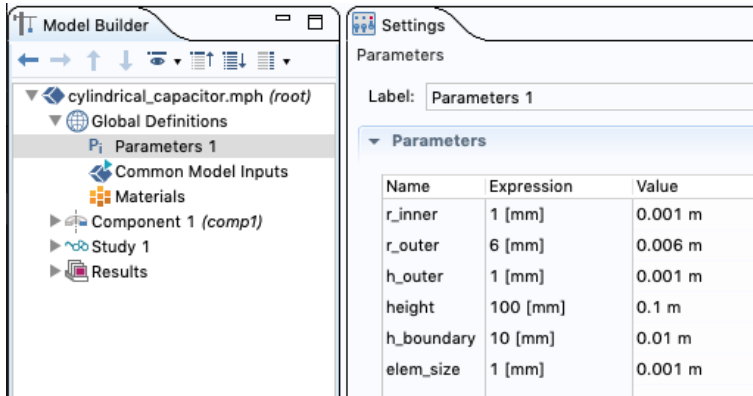
1.2. select *Blank Model*

1.3. Comsol opens a new project. The layout of the Comsol main window is shown below.

1.4. In the model builder we select the node termed *Parameters*. Here we can define all parameters relevant to the setup, e.g., the geometric distances and the mesh element size.

1.5. In the settings window add the parameters as shown in the screenshot. After the numeric value you should add the unit in brackets. Comsol will convert everything into SI units and it will perform unit checking whenever you enter expressions in COMSOL.

Name	Expression	Value
r_inner	1 [mm]	0.001 m
r_outer	6 [mm]	0.006 m
h_outer	1 [mm]	0.001 m
height	100 [mm]	0.1 m
h_boundary	10 [mm]	0.01 m
elem_size	1 [mm]	0.001 m

1.6. We need to add a component to the model. According to the COMSOL manual:

> "A model component is a fundamental part of the model and contains a geometry with its associated physics interface, mesh, and variables and other definitions that are local to that component. The Component node defines the namespace for each part of a model that is defined in a model component."

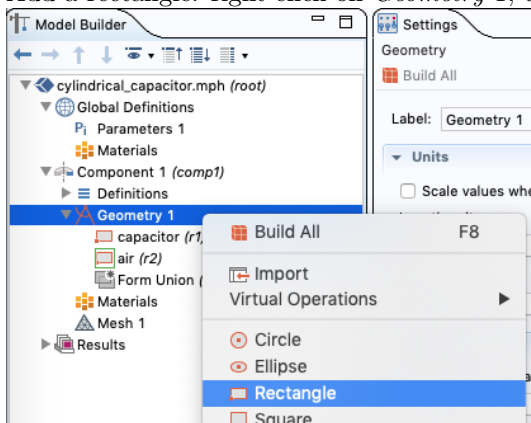Right click on the root node of the model. Select *Add Component 1* and then *2D Axisymmetric*.
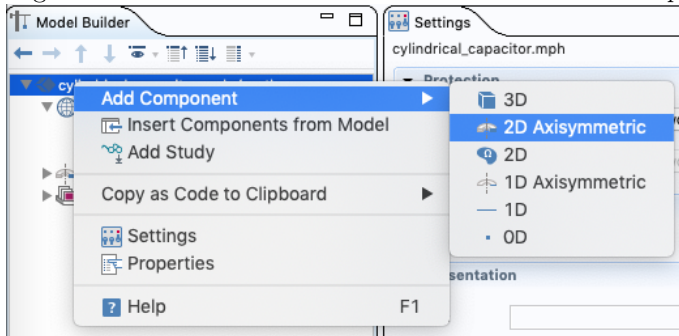
The component appears as a new node with some default child nodes.

Geometry

Next, the geometry needs to be modeled.

1.7. Add a rectangle: right click on *Geometry 1*, then select *Rectangle*.

1.8. In the settings window, name the rectangle *capacitor*. Set the *Width*, *Height* and *Position* using the previously defined parameters (see 1.5.). Disclose the *Layers* section and add a layer with thickness `r_inner` to the left and right sides. Note: We are making use of the fact that `r_inner = h_outer`. Otherwise, you would need to create two rectangles instead.
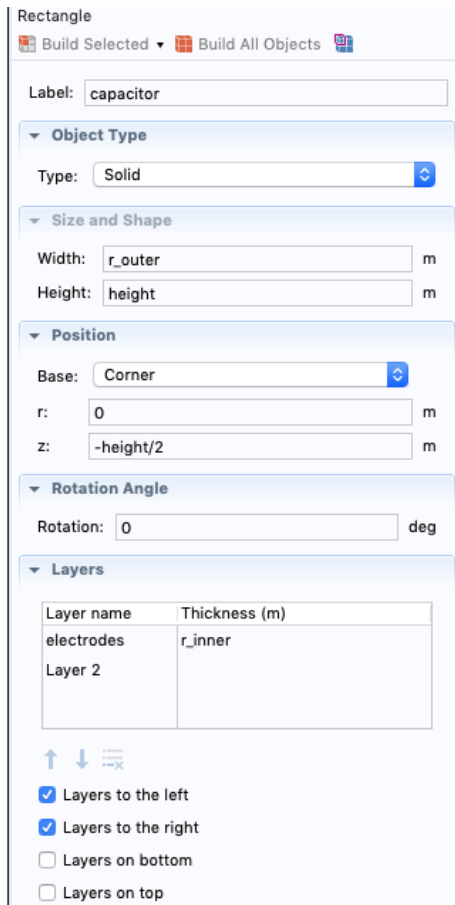
1.9. At the top of the settings window you can click on *Build All Objects* to have a look at the resulting geometry.

1.10. Add a second rectangle and name it *air*. The *Width* should be `r_outer+h_boundary` and the *Height* `height+2*h_boundary`. *Position* the rectangle at r = 0 and z = `-height/2-h_boundary`. Add a *Layer* of thickness `h_boundary` to the bottom and top. Note: the layers are not really required here and you could leave them out.

1.11. Press *Build All Objects* again. The resulting geometry should look as in Fig. 1.

We need to assign appropriate materials to each of the domains in the geometry.

1.12. Right click on *Materials* and select *Add Material from Library*. The material library browser pops up.
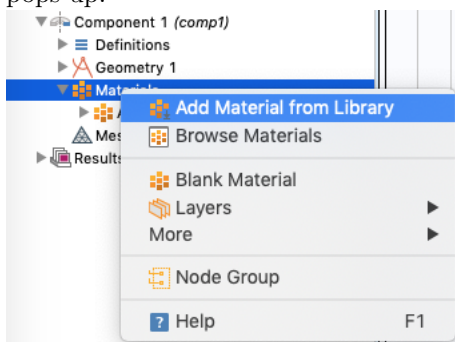
1.13. Go to *Built-In* and double click *Air* to add it to our component (alternatively: right click *Air* and select *Add to Component 1*). Similarly add *Built-In → Aluminum* and *MEMS → Insulators → Si3N4-Silicon nitride*.
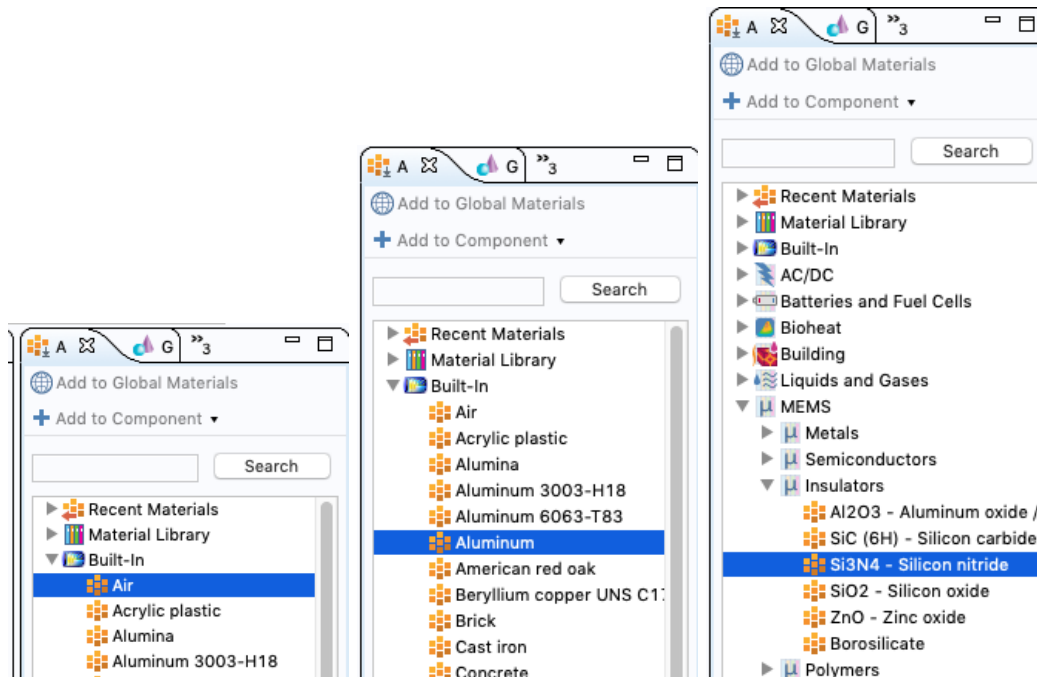
1.14. Three new nodes appear under the *Materials* node in the model builder. Select the *Air* node and in the settings window make sure that *All Domains* is selected in the *Geometric Entity Selection* section. This assigns air to the whole simulation domain.

1.15. Next, select the *Si3N4 - Silicon nitride* node. Under *Geometric Entity Selection* choose *Manual*. You can now click on the domain between the two thin electrodes to assign silicon nitride to it.
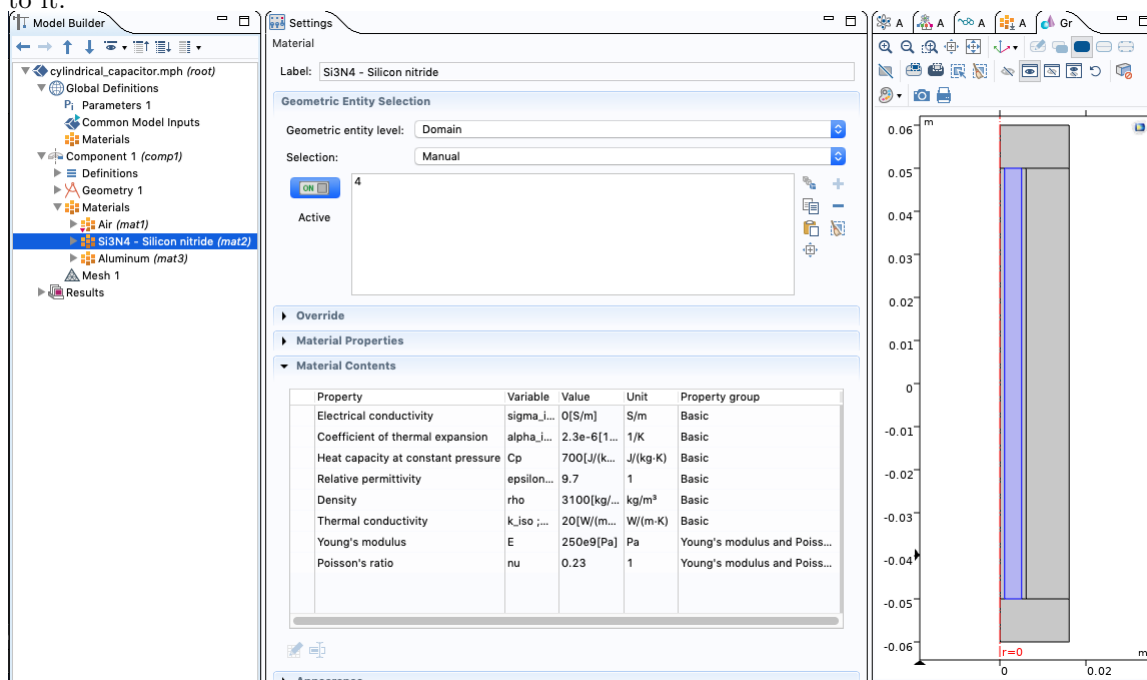
1.16. The tree of nodes in the model builder is hierarchical. A node overrides any conflicting settings of all nodes above. Notice the small down-pointing red arrow in the icon of the *Air* node. It indicates that the currently selected node (partially) overrides properties of the *Air* node. If you go back and select the *Air* node, you will notice that the graphics window now indicates all geometric domains except the Si3N4 region to consist of air (blue regions). Additionally, you will find a small red upward-pointing arrow in the icon of the *Si3N4* node. This indicates that the currently selected node is being (partially) overridden by that node.

1.17. Similarly, select the *Aluminum* node and assign it to the two electrode regions by selecting them with the mouse. This further overrides parts of the *Air* node specifications, leaving air only in the surrounding region.
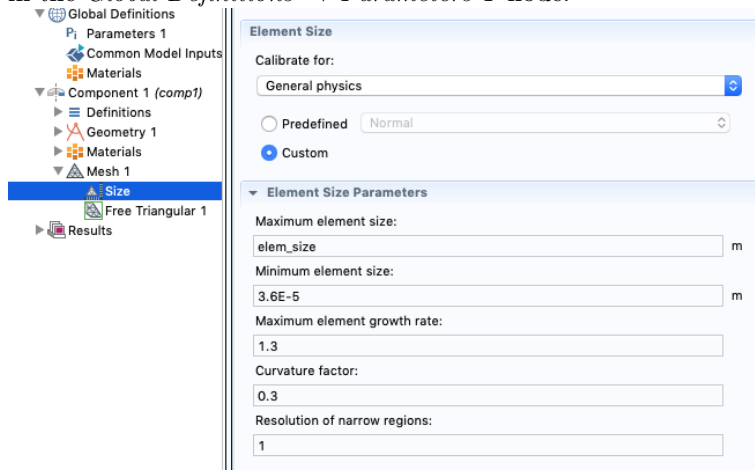
Meshing

The finite element computation requires a meshed geometry. This means that the geometry should be divided into small geometric entities, e.g., triangles, on which the partial differential equations describing the physics can be solved. The meshing can be performed in COMSOL with the following steps:

1.18. In the model builder select the node named *Mesh 1*.

1.19. In the settings window select *User-Controlled mesh* as the *Sequence type*.
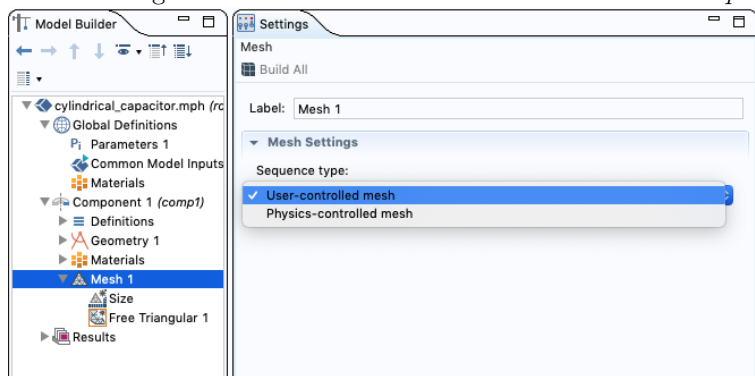
1.20. Disclose the *Mesh 1* node to show its children.

1.21. Select the *Size* node and select *Custom* as *Element Size* in the settings window.

1.22. Specify the *Maximum element size* using the parameter `elem_size` which we already defined in the *Global Definitions → Parameters 1* node.

1.23. Note the *Free Triangular 1* node that has already been created by default. It means that the geometry will be meshed with triangles.

1.24. In the settings window click on *Build All*. This triggers the meshing process. Upon finishing, the graphics window shows the resulting mesh. An excerpt of the top part is shown below.

6
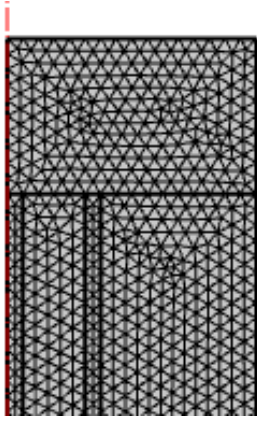
Note: Instead of the default *Free Triangular* mesh, you could use a *Mapped* mesh by right clicking the *Mesh 1* node and selecting *Mapped* (and deactivate/delete the *Free Triangular 1* node). This would create regular quadrilaterals wherever possible. It is sometimes preferable to use a regular mesh.

2 Part II. Simulation

In Part I we created a model of the geometry and assigned material parameters to each part of the setup. Additionally, we discretized the geometry by meshing the domains with small triangles. However, the physical laws governing the behavior of this setup are still missing. We will use partial differential equations (PDEs) and boundary conditions (BCs) that have already been predefined in COMSOL for this purpose. In COMSOL's terminology these are called *Physics* and *Boundary Conditions* respectively.

2.1. Right click on the *Component 1* node and select *Add Physics*.
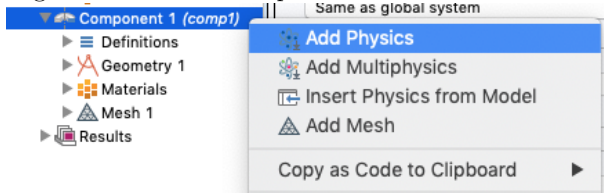
2.2. The physics navigator window opens. Disclose the *AC/DC* section and double click on *Electrostatics (es)* to add the corresponding PDEs to our component.
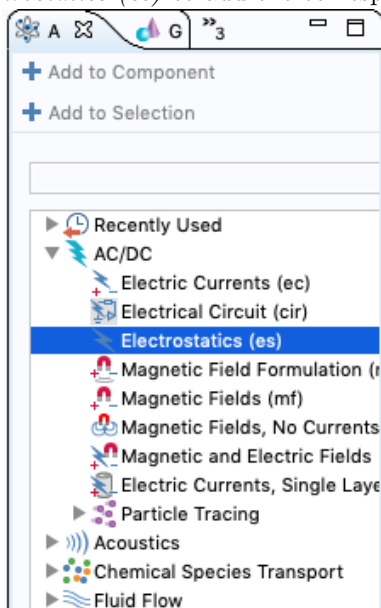
2.3. Select the new *Electrostatics (es)* node in the model builder to inspect the settings. Note that per default the electrostatic field will be calculated on the whole domain of the previously modeled setup.
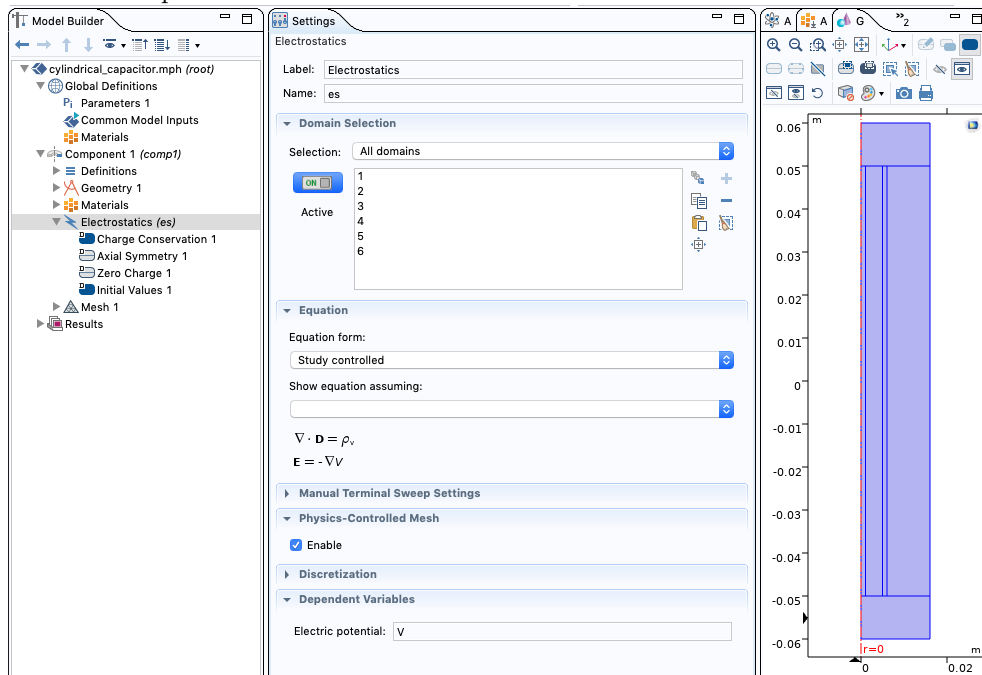
You will also be able to see the PDEs that will be solved by disclosing the section *Equation* in the settings window. This section is hidden by default. To activate it, go to the top of the model builder window and click on the eye. Then select *Equation Sections*.
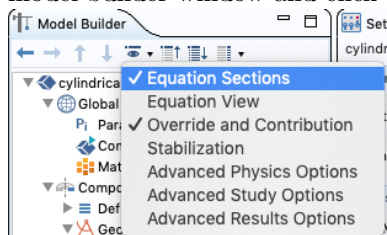

2.4. Some default child nodes have also been added to the *Electrostatics (es)* node. Disclose it to have a look at them. The first one is the *Charge Conservation 1* node, i.e., $\nabla \cdot \vec{D} = \rho_{\mathrm{f}}$, where $\vec{D}$ is the electric flux density and ρ_{f} is the free charge density. This is correctly applied to the whole simulation domain per default.

2.5. There are two default BCs that have also been added. The first one is given by the *Axial Symmetry 1* node. COMSOL has correctly inferred this BC from our axially symmetric geometry that we have modeled. By selecting it, you will see that it is being applied only to the axis of rotation.

2.6. The second BC is represented by the *Zero Charge 1* node. Click on it to see that it is being applied to all boundaries except the axis of rotation. The zero charge condition is $\vec{D} \cdot \vec{n} = 0$, where $\vec{n}$ is the outward pointing normal vector to the boundary. This means that the electric flux $\vec{D}$, and hence also the electric field $\vec{E}$, at the boundaries will be forced to be parallel to the them.
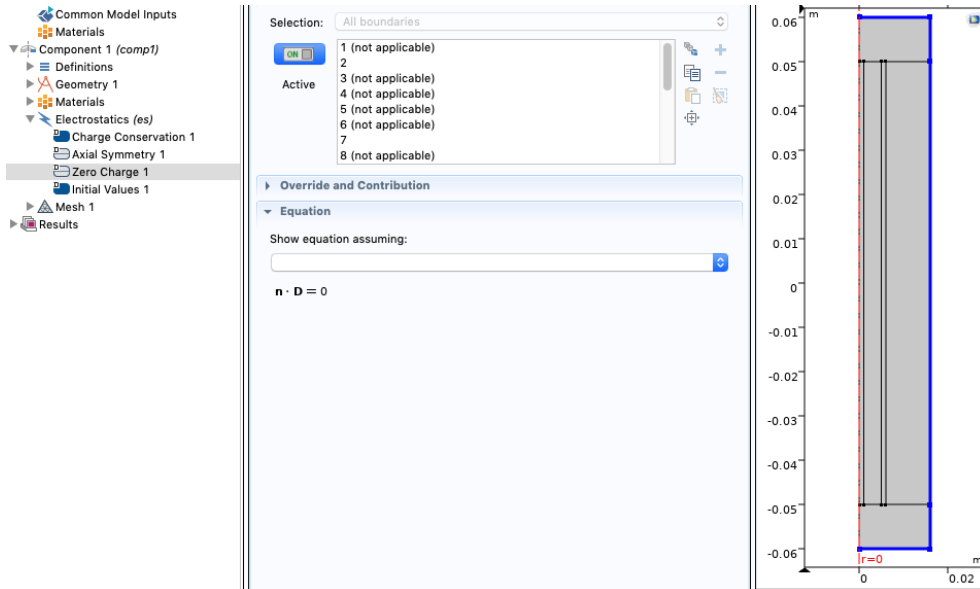
2.7. The *Initial Values 1* node specifies that the electric potential V is zero on the whole domain. Note that the default nodes are marked with a "D" in the icon of the nodes. These cannot be deleted, but only changed or overridden, as they specify a minimal set of conditions required to obtain a well posed problem.

2.8. For the problem we want to model, we need to add two more constraints on the electrode domains specifying a uniform and constant desired potential. Right click on the *Electrostatics (es)* node and select *Terminal*. Note that the options are grouped by whether they operate on domains (filled rectangle) or lines (not filled).
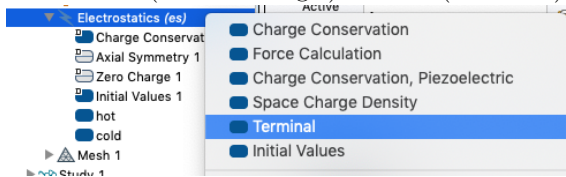

2.9. Click the *Terminal 1* node and label it "hot". Select the inner electrode as domain. Apply a voltage of 1 V.
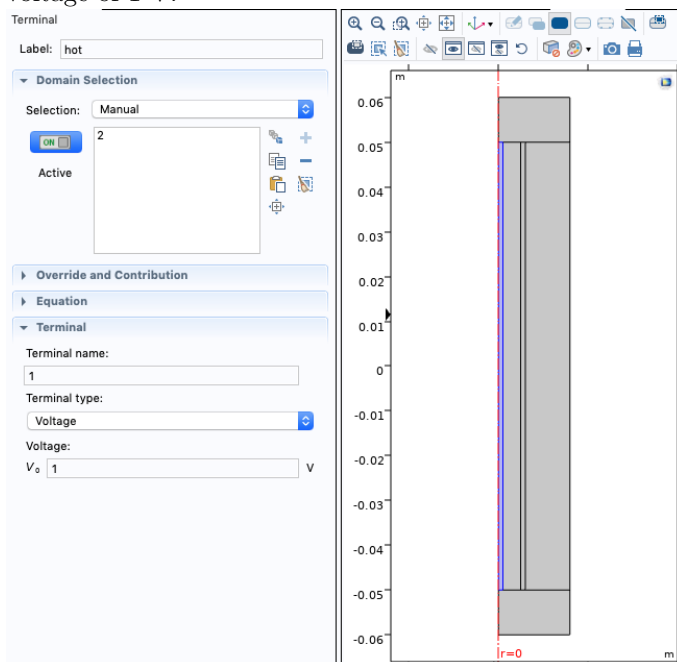

2.10. Add a second terminal node, label it "cold" and apply it to the outer electrode domain. Leave the potential at 0 V.

9

This completes the setup of the physical model. Lastly, we have to specify what kind of simulation to perform and what numerical procedures to use in order to solve the underlying linear system of equations. This is specified in COMSOL in a so called *Study*.

2.11. Right click on the root node of the model and select *Add Study*.
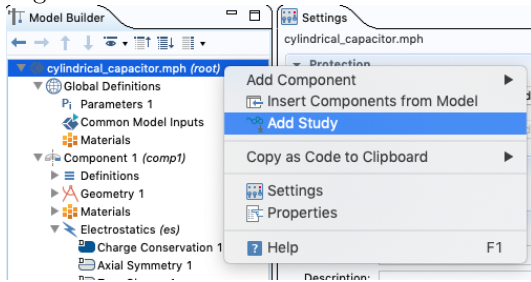
2.12. In the navigator double click *Stationary* to add such a simulation to our project.
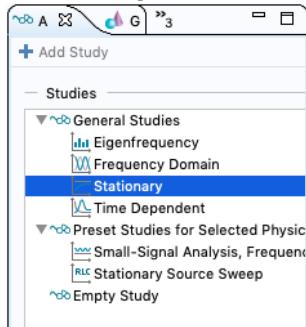
2.13. By disclosing the *Study 1* node you will find one child node (depending on your settings maybe more). Any desired settings could be performed here. Leave the defaults for now.
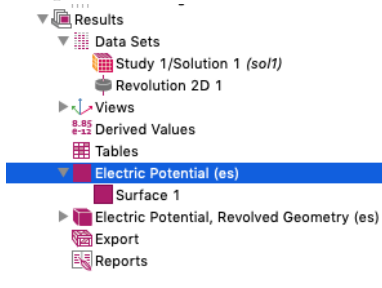
2.14. In the *Step 1: Stationary* child node, click on *Compute* to trigger the simulation.
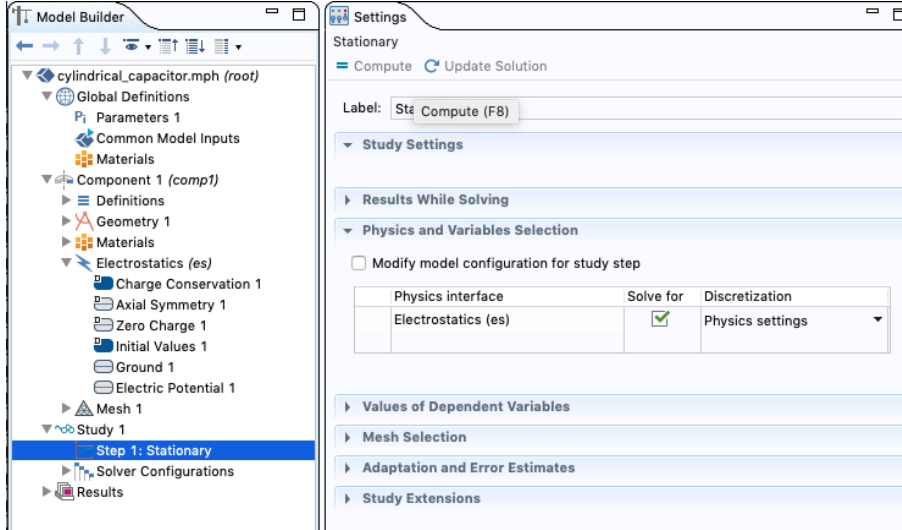
3 Part III. Postprocessing and Visualization

After computation, some default plots are generated. We inspect them and also create some custom plots.

**Electric potential:** From the *Results* node, select the child node named *Electric Potential (es)*.

The Graphics window shows a surface plot of the calculated electric potential. Under the *Data* section, note that the selected *Data set* is the solution of the *Study 1/Solution 1 (sol1)*. This indicates that the solution of the simulation we just performed is being plotted.

To obtain a better visual representation we change the color table. Select the child node called *Surface 1*. Under *Coloring and Style*, select the *ThermalEquidistant* color table and activate the *Reverse color table* option. The result should look as on the right side.
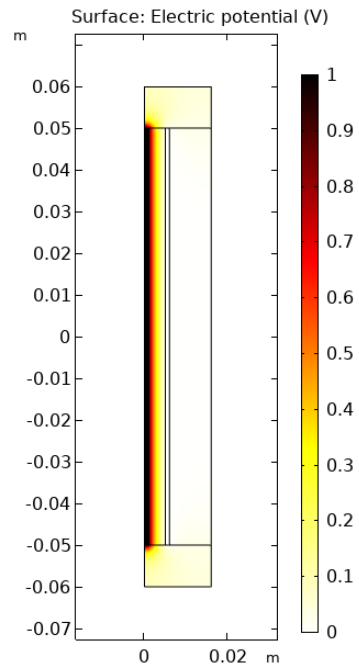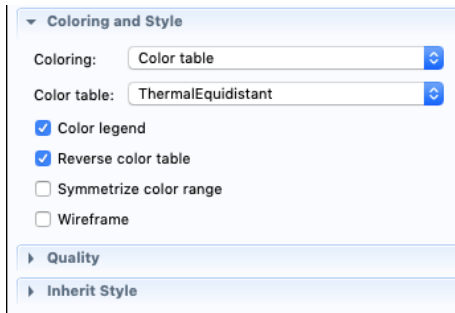

Electric potential revolved: We have computed the electric field on a reduced 2D geometry of a setup which actually exhibits rotational symmetry. The results of the whole 3D setup can be extrapolated and visualized. COMSOL has already done this per default. Select the node called *Electric Potential, Revolved Geometry (es)* to see the electric potential in 3D. Under the *Data* section, note that the selected *Data set* is a different one. It is stated as *Revolution 2D 1*. Going to *Results → Data Sets* we find the node which contains the simulation results, i.e., *Study 1/Solution 1 (sol1)*, as well as the data set obtained by revolution of this solution, i.e., *Revolution 2D 1*. If required, more data sets could be added here by performing different kind of operations on the original solution. You can inspect the possible operations by right clicking on the *Data Sets* node.
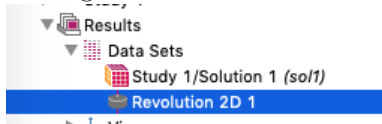
Select the *Revolution 2D 1* node. In the settings windows change the *Revolution angle* to 210°. Go back to the *Electric Potential, Revolved Geometry (es)* node to see the changed plot of the revolved electric potential.
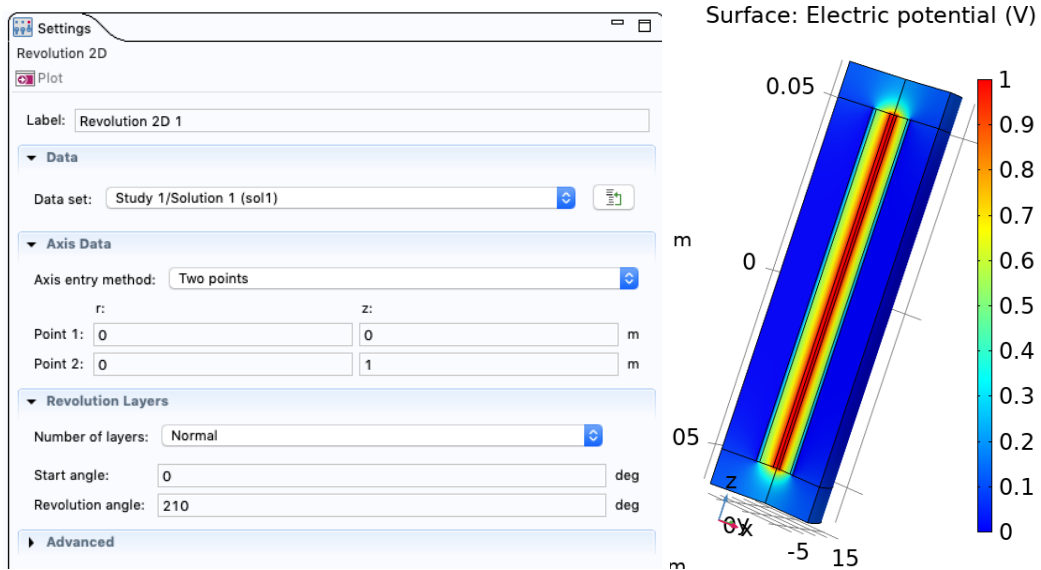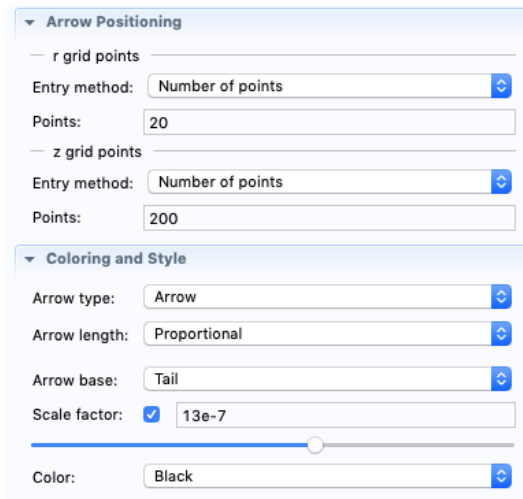

Electric field: The electric field intensity $\vec{E}$ is a vectorial quantity. In order to visualize it we can plot the vectors at selected points in space. Right click on *Results* and select *2D Plot Group* to add a new plot group with two axes. The newly created node can contain several plots as sub-nodes which will be plotted into the same coordinate axes. To begin, label the node as *Electric Field*. Then right click on the node and select *Arrow Surface*.
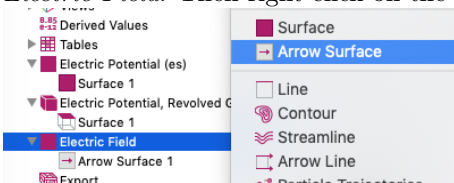

In the *Expression* section we can specify what we want to plot. If you know the variable names used by COMSOL, you can write them into the text fields. Otherwise, click on the button with a green and red triangle on the right side and select the variables you want to plot.
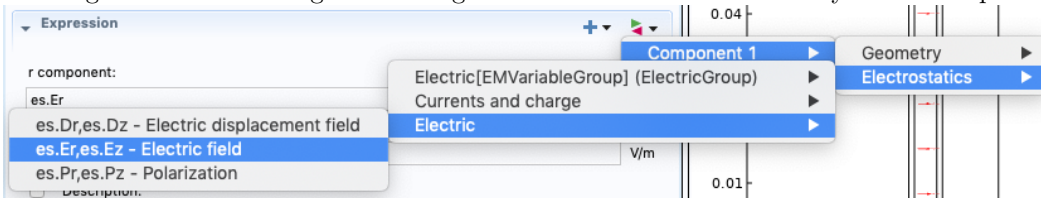
It is possible to specify the number of vectors that are plotted at selected positions of the radial and axial coordinates. This can be found under *Arrow Positioning*. Lastly, the arrows should be scaled so that they do mostly not overlap. Go to the section *Coloring and Style* and specify a scale of `13e-7`. You might need to experiment with this value, as well as with the *Arrow Positioning* settings.

Lastly, we can also add the equipotential lines to the same plot. Right click on the *Electric Field* node and select *Contour*. Leave the default settings and click on *Plot* in the top of the settings window. This will plot the electric potential contour lines.


Electric field vectors and equipotential lines
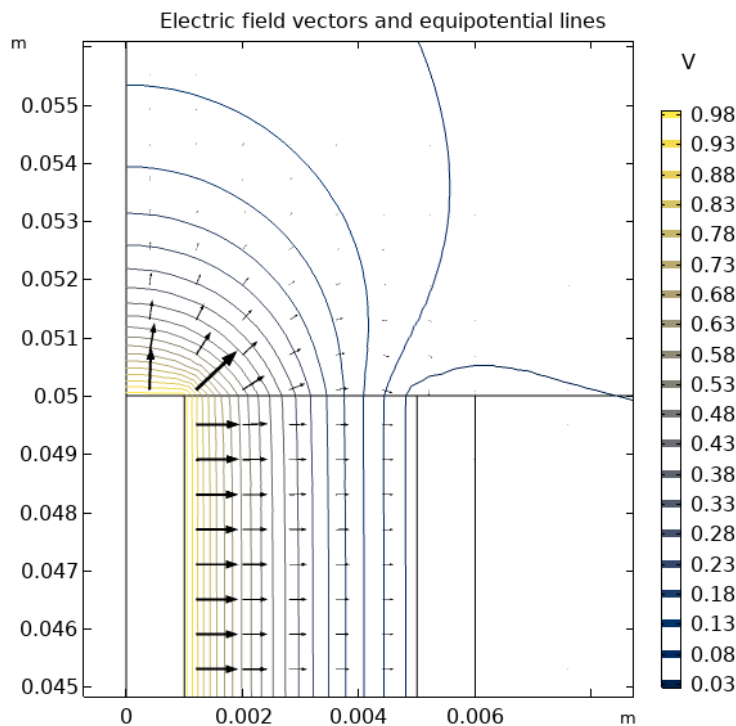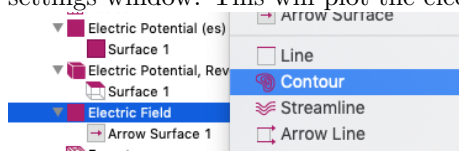
Capacitance: The computed field is a detailed space-dependent solution of the problem. It is often convenient and sufficient to consider only some sort of "global behavior" of the system.

Network theory considers the capacitor as a *lumped element*, requiring only one numerical value, namely the *capacitance C*, to describe the electrical behavior of the system. The capacitance is defined as the amount of charge Q required to increase the voltage U by 1 V, i.e., $Q = CU$.

In the following, we analyze the convergence of the capacitance with respect to the size of the surrounding air domain. As the field is only non-zero close to the capacitor, it is understood that only a region close to the capacitor needs to be modeled. But how close? To answer this question we can compute the problem for different values of the air domain boundary `h_boundary`. This requires us to regenerate the geometry and the mesh and then compute the

13

solution for every value of `h_boundary`. COMSOL can automatically perform these tasks for us.

3.1. Right click on the *Study 1* node and select *Parametric Sweep.*
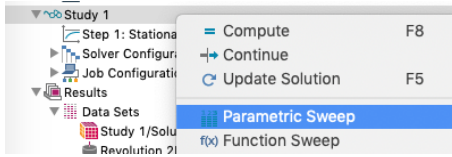
3.2. In the *Study Settings* section click on the plus sign and select `h_boundary` as sweep parameter.

3.3. Click on the range symbol (the last symbol in the tool bar below).

3.4. In the pop-up window, specify values for start, step and stop and then click replace.
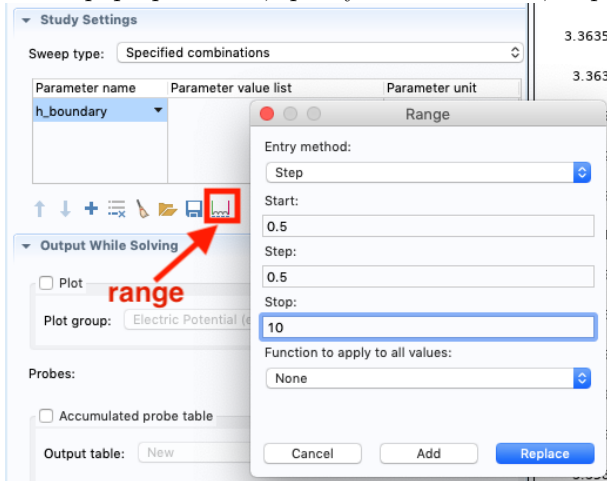
3.5. Change the *Parameter unit* to mm.

3.6. At the top click on *Compute* and wait for the process to finish. You can check the status at the lower right of the main window.

3.7. Note that after the computation a new *Data Set* has been created called *Study 1/Parametric Solutions.* This data set can be used for plotting the results of the parameter study.

3.8. Right click on *Results* and select *1D Plot Group.* Name the new node "Capacitance over boundary size".
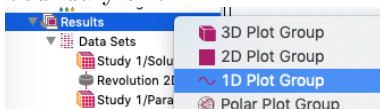
3.9. In the *Data* section of the settings window choose *Study 1/Parametric Solutions.*
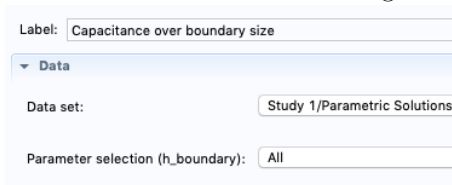
3.10. Leave the *Parameter selection* option at the default value of *All.*

3.11. Right click on the group plot node that you just created and select *Global.* This kind of plot is used to access all global properties, e.g., total stored energy or capacitance.

3.12. In the *y-Axis Data* section in the settings window, click the variable selector icon and choose the *C11 Maxwell capacitance.*
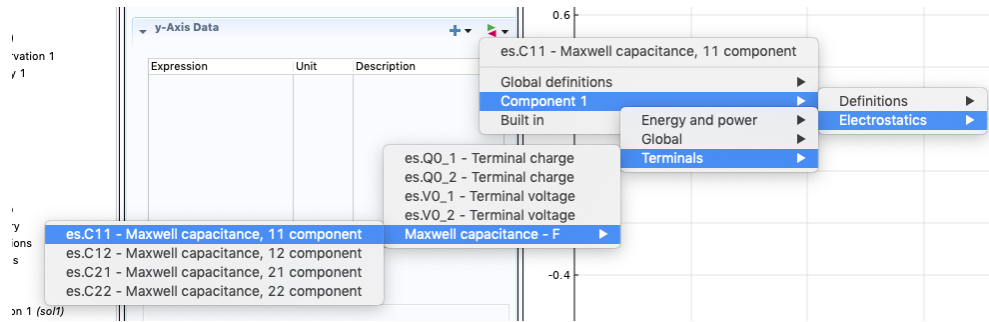
Note that C11 refers to the capacitance of "terminal 1", i.e., the node *Component 1 →
Electrostatics (es) → hot*, which has the *Terminal name* 1.

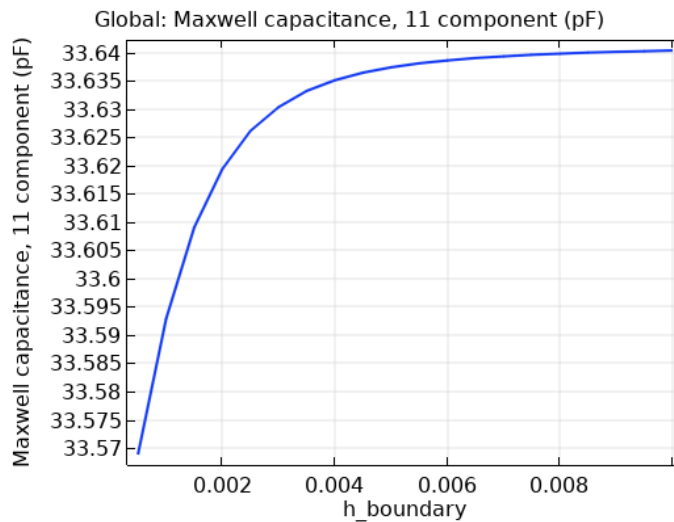3.13. Change the Unit to pF.

3.14. At the top of the settings window, click on *Plot*.

3.15. Note the exponential convergence of the capacitance with increasing boundary size. From
the above plot you can see that a boundary of more than 5 mm yields good results.

3.16. If necessary, you can change the labels of the axes in the settings window of the *1D Plot
Group* in the *Plot Settings* section.
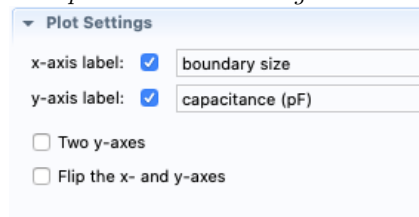
3.17. If you have more than one curve in the plot you need to add a legend. You can do so by
activating it in the settings window of the *1D Plot Group* and entering the corresponding
label in the settings window of the *Line Graph*.
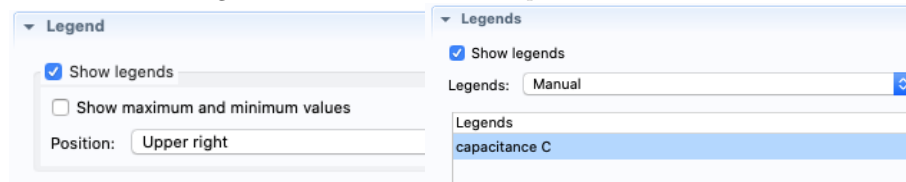
3.18. Sometimes it is preferable to perform the plotting in another program, e.g., MATLAB,
gnuplot or python. You can export the data of the plot by right clicking on the *Export*
node at the very bottom of the *Model Builder*. Select *Plot* and configure the settings
window as is convenient for you. Choosing *Text* as *File type* yields a conventional text
file that can be imported easily in MATLAB or other plotting software.

15

4 Part IV. Validation

A FE model is always an **idealized** version of a real setup. The person modeling the problem is required to have a good understanding of the real physical situation and how to approximate it with the equations, boundary conditions and loads available for computational purposes. Many a-priori assumption are made hereby. It is, therefore, substantial to validate the FE model against known solutions and properties (e.g., convergence) of the model.

In the last section we already validated the choice of air boundary size by inspecting its influence on the capacitance of the model. In general, global properties of the system are well suited to check the convergence of the solution with respect to model parameters such as element size or air boundary. They are also very convenient to validate the simulation against analytical solutions. Two very important validations that should be performed with every model are presented below.

Comparison to a similar analytical solution: The analytical solution of a cylindrical capacitor is well known. The capacitance can be computed as

$$C = \frac{2\pi \varepsilon l}{\ln b/a}, \tag{1}$$

where ε is the permittivity of the dielectric material, $l = $ `height`, $b = $ `r_outer-h_outer` and $a = $ `r_inner`. **Task:** Compare the analytical solution of the above setup to the capacitance obtained with the FE computation. Are the two solution in good agreement? Keep in mind, that the analytical solution ignores the stray field at the top and bottom of the setup and is, hence, an approximation. On the other hand, the FE model might be prone to discretization errors (which can be analyzed as shown in the next section).

Mesh study: The FE method solves BVPs by discretizing equations in space and time (for transient problems). It is, therefore, necessary to check the convergence of performed computations. A very important concept is a *mesh study*. This refers to the computation of the problem with differently fine meshes. When the solution does not change much between meshes with different element sizes, convergence of the solution has been obtained. This is based on the idea that the solution of the problem should be independent of the mesh, as it is merely a concept of the solution procedure and not of the problem itself.

In COMSOL you can perform a mesh study with a *Parametric sweep* on the `elem_size` parameter. Usually, the total stored energy is then plotted over the `elem_size` to check convergence. This gives a good insight into the convergence behavior of FE simulation you set up. **Task:** perform the mesh study as described above and plot the result.

5 Final Remarks

Sharing your model

Everything relevant to the FE model is stored in the `.mph` file. This includes the parameters, geometry, mesh, PDEs, BCs, solutions and plots. Especially the solutions and the mesh can become very big. Before sharing your model, make sure to delete the mesh and the simulation results.

- Go to *Edit* → Clear All Meshes.

- Go to *Edit* → Clear All Solutions.
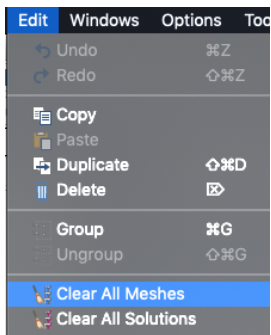
Note that after this operation the meshing and the computation will need to be performed again. For big models, this might be a time consuming process. Nevertheless, all information required for these two steps is still contained in the .mph file.

Conclusion

A model for a cylindrical capacitor has been setup from scratch and the calculated electrostatic field has been visualized. We additionally inspected the influence of the air domain on the capacitance of the structure and discussed the importance of model validation.