# Lab X: Electrostatics with COMSOL Multiphysics

In this week's lab, you'll gain familiarity with COMSOL Multiphysics, a common software tool used to simulate physical processes by scientists and engineers in academia and industry. COMSOL can simulate basic mechanics, thermal physics, fluid dynamics, and electromagnetism, and can combine these processes together, so you could, for instance, study the air flow around an electric motor as it heats up. But today we'll be using it to perform an electronic version of the Coulomb's Law experiment.

Lab 2, Coulomb's Law, has an important limitation: we measure the force between two charged spheres, but we don't know the charge on these spheres: we only know their electric potential, aka voltage. Even worse, when the spheres are close together, the charge on each one is repelled from the other, so the charge is unevenly distributed in ways we can't measure experimentally. Lab 2 contains some calibration constants and correction factors to deal with these problems. How were those correction factors obtained?

One way to do it is by solving the problem using a computer. Of course, we can't use a computer simulation to prove that Coulomb's Law is true, or to derive the Coulomb constant, because the simulation *assumes* it's true. Nevertheless, this can help us make the experimental work more precise.

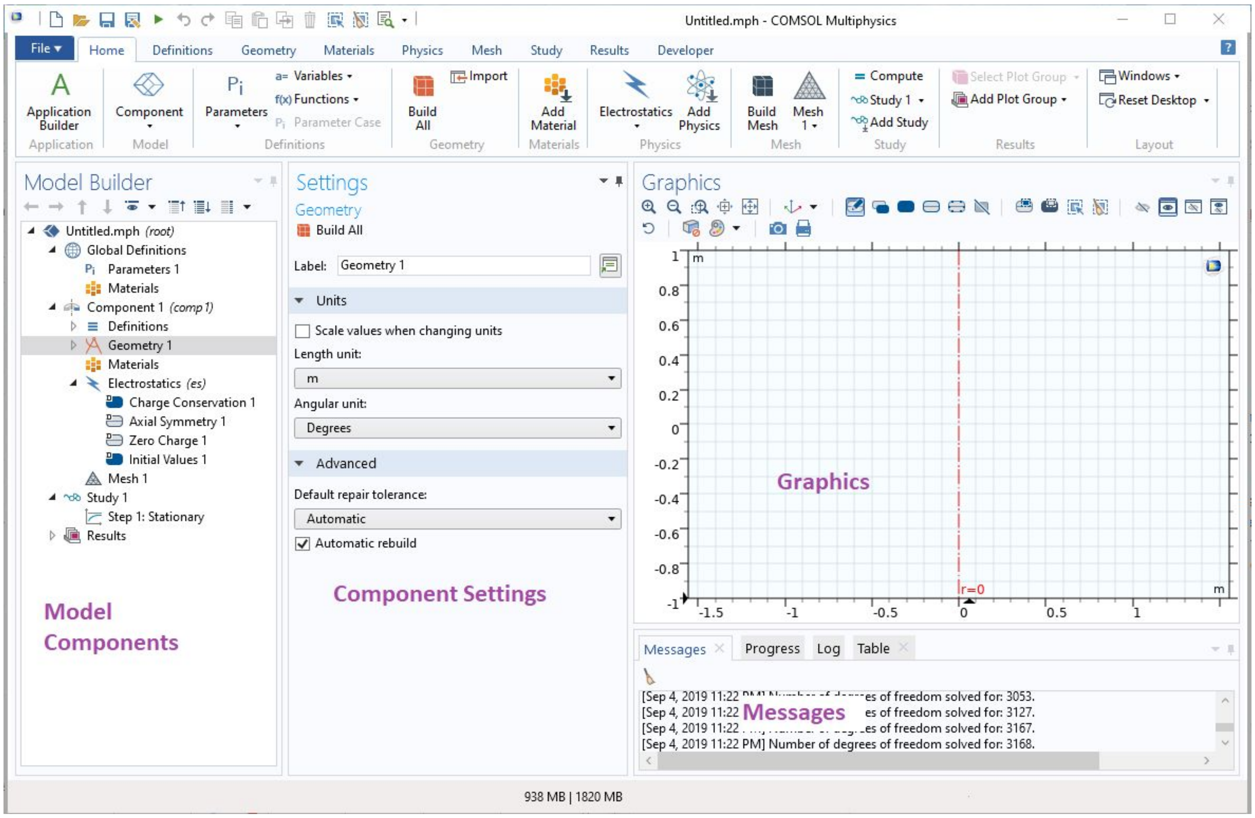
Here's our problem: we will have two spheres of radius $a$ = 1.9 cm, separated by a variable distance *"sep"*. We will charge each sphere up to 3000 volts, and measure the electric force on one of them as a function of *sep*. We will find that the force-vs-distance doesn't <u>quite</u> obey an inverse-square law because the charge is distributed evenly over the spheres. By how much does it deviate?

## Procedure

### Picking the Model Type

1. Start COMSOL Multiphysics on the laptops provided.
2. Choose the option to create a new model using the "Model Wizard".
3. Next you'll see a list of possible space dimensions. For this experiment, choose "2d axisymmetric".
4. Next you'll see a list of possible kinds of physics that can be included. Choose the "AC/DC" list, and pick "Electrostatics (es)". Click "Add" to add these physics equations to the model. On the right, you can see that the only relevant physical variable calculated by this physics interface is the electric potential V. (If you wanted to calculate currents, magnetic fields, etc, you'd need to add other physics interfaces.) Now choose "Study".

5. COMSOL can study the equilibrium “stationary” behavior of a system, or how it varies over time, either in response to transient changes or how it responds to a periodic forcing. For this study, we want a “Stationary” study. Select it, then click “Done”.
6. You’re now presented with the main user interface for COMSOL.

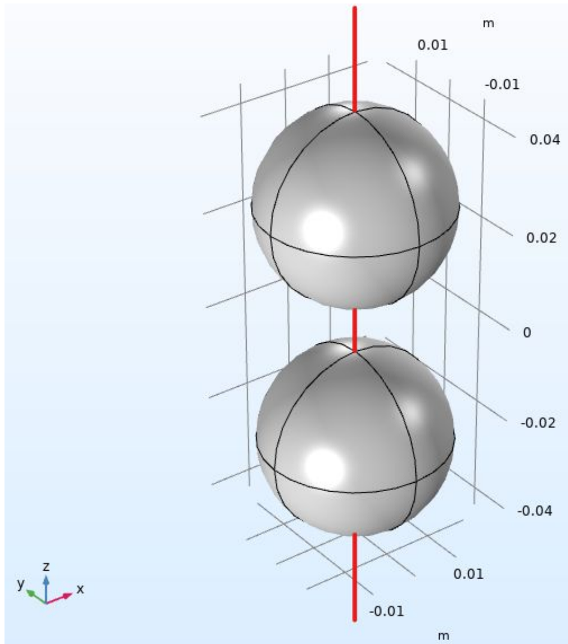


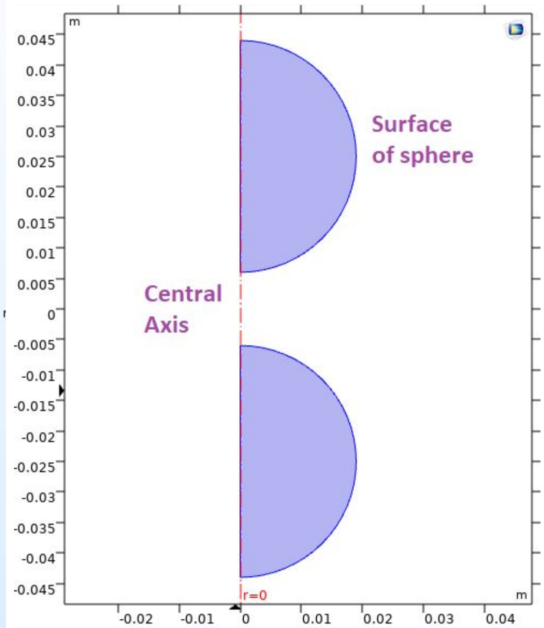

The layout looks a little different between Mac and Windows, but the key elements are the same. The “Model Components” section on the left, also called the Model Builder, shows all the pieces you’ve added to the model: geometric shapes, material properties, boundary conditions, model physics, or graphical outputs. To add a new piece, you **RIGHT-CLICK** (control-click on Mac) on the kind of component you want to add. The “Component Settings” section lets you fine-tune each of the elements you’ve added to the model. The “Graphics” area shows charts and graphs, and the “Messages” section gives text feedback.

7. Try left-clicking on the “Electrostatics” section of the Model Builder and click the triangle next to the “Equation” tab. You should see the equations COMSOL is solving. (What COMSOL calls ***D*** is almost the same thing as the electric field ***E***.)

## Setting Up Objects

8. Our model geometry will look like the left-hand picture below in 3-d, but since it’s symmetric about the *z* axis, we’ll be working with a *cross section view* (seen on the right). It’s faster to compute, easier to set up, and more accurate.



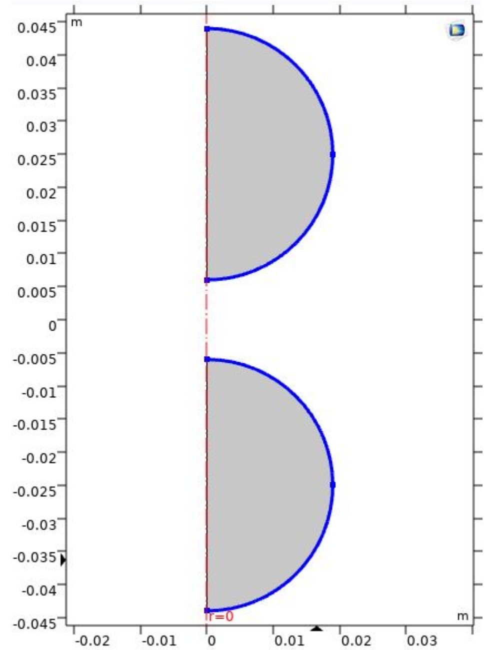




9. First, let's define some key parameters to describe the separation between spheres, which will vary, and their radius, which will not.
    - Left click on "Parameters 1". In the "Name" column of the table, type "sep". In the "Expression", type "5 [cm]". The square brackets indicate units. COMSOL cares a lot about units, and you should too! Add a useful description.
    - In the next row of the table, create a parameter called "a", for the radius of the spheres, with a value of 1.9 cm.
10. Next, let's create our upper sphere, whose cross-section is a circle.
    - Right click on "Geometry 1" and choose "Circle" to create a new object. In "Settings", set its Label to "Upper sphere", its Radius equal to "a", its r-position to zero, and its z-position equal to "sep/2". Click "build all objects" to see your circle.
    - Create a second circle for the lower sphere the same way, with the same values except its z-position should be "-sep/2". "Build all objects" and you should see both circles now. If you don't, click the  above the graphics window to adjust the view to show all objects.

11. Next, we need to tell COMSOL what material our spheres are made out of. While their surface is conductive, inside they're lightweight foam which is basically air. Right click on "Materials" under "Component 1" (not under "Global Definitions") and choose "Add Material". COMSOL includes a vast library of pre-defined materials. From "Built-In", double-click on "Air". Both of our objects should light up blue, and the "Geometric Entity Selection" should say that "All domains", including objects #1 and #2, are made of air.

The lower section of the Materials Settings panel shows what properties of air are needed by this physical model. In this case, just the “relative permittivity”. We’ll discuss what that is later.

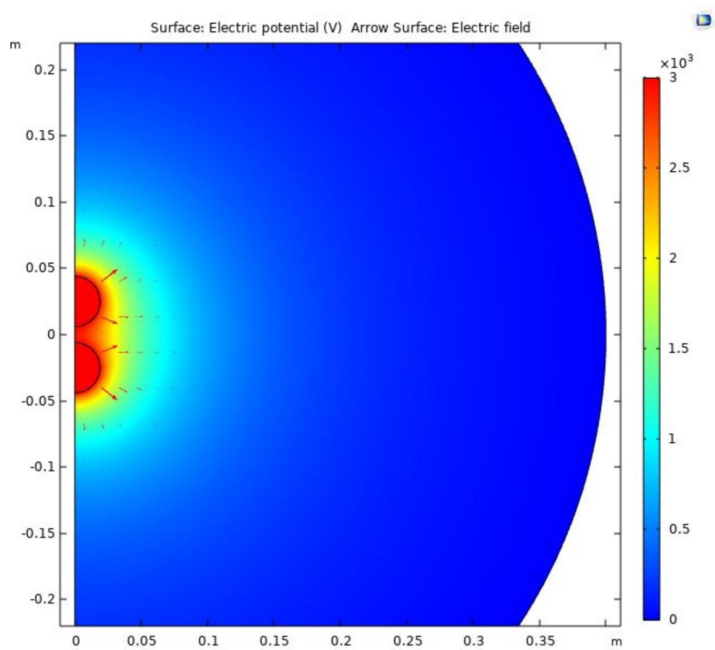
12. Next, we need to define boundary conditions for our model physics. We will assume that both spheres are at a potential of 3000 volts.
    • Right click on the “Electrostatics” section of the Model Builder and choose the “Electric Potential” boundary condition. Now click on the curved outer surface of both objects to highlight them, as shown at right. Do not highlight the center line!




13. Are we done? Well, almost. We need to calculate the electric field and potential not just on the plates, but the space between and surrounding them. So we need a third object to represent this region. Create a third circle, labeled “Empty Space”, with a radius of 50 cm and its center at r=0, z=0. Rescale the view with to see it. Click on the “Air” material to check that this new circle (well, sphere actually) is also made of air.
14. Finally, we need to tell COMSOL that the voltage should go to zero as we go very far from the two spheres… which in this case means V = 0 on the outer edge of the “Empty Space” object. Right click on “Electrostatics”, choose the “Ground” boundary condition, and click on the outer edge of the “Empty Space” circle. Don’t click on the center line.

**Running the Model**

15. COMSOL works by breaking each object up into a bunch of triangular pieces, and solving the E&M equations on each piece. Click the “Mesh 1” item from the Model Builder, and choose “Fine” from the Element Size dropdown to create smaller triangles for a more accurate result. Then click “Build All” to see the mesh COMSOL is using.
16. Now, let’s calculate the results! Click on Study 1, choose “Compute”, and wait a few seconds. COMSOL will show you a pretty color map of the electric potential



(voltage) in this domain. It should be 3000 near the two spheres and zero far away.

17. Let's take a look at the electric field near the spheres. The items in the in the Results tab on the left-hand column show various views of our computation. Right-click on the "Electric Potential (es)" view and add an "Arrow Surface". COMSOL correctly guesses that the vectors we care about are the electric field.

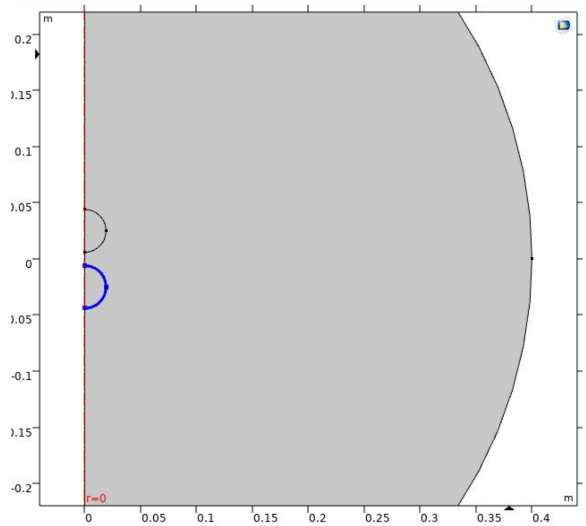
**Analyzing the Results**

18. Remember, our main job is to calculate the force on one of the spheres. If you're in the second group to do this experiment, you will recall that the force on a charged object is

$$\vec{F}_e = q\vec{E}$$

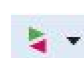
When the charge is distributed over a surface, we must add up the electric forces on an infinite number of tiny surface area components *da* over the surface S:

$$\vec{F}_e = \int_S \sigma \, \vec{E} \, da$$




where $\sigma$ is the charge per unit area. If you're in the first lab group, we haven't covered this yet, but either way, don't panic! Even though $\sigma$ varies and the shape is a sphere, COMSOL can do this pesky integral for us.

19. In Results, right-click on "Derived Values" and choose "Integration / Line Integration". (in our cross-section view, the surface of the sphere is a line.) Select the round edges of *only the lower sphere*, as shown in the picture at right.
20. Let's integrate two quantities. First, integrate the charge density $\sigma$ times the z-component of the field to get the z-component of the electric force. In the Expressions box, type "es.nD * es.Ez" -- the first term is COMSOL's variable for surface charge density, the second is the z-component of E.

    (You can get a complete list of possible variables by clicking the  drop-down.) Set its description to "Electric Force".
21. Now, click the "Evaluate" button at the top. A table should appear below the graphics window with the results of the calculation.
22. Write down the force and separation in a Google spreadsheet.
23. Repeat the experiment with separation values of 5, 6, 8, 10, 15, 20, and 30 cm. Record the force for each. You will need to re-do the "Study 1/Compute" and "Line Integration 1 / Evaluate" commands each time.

**Question:** If the spheres could be modeled as infinitely small points obeying Coulomb's law, the force would be inversely proportional to $1/sep^2$. That is, a graph of 1/sep^2 vs force should be a straight line. Is it?

**Part 2: Parallel disks**

I will show you an apparatus composed of two parallel disk-shaped plates. Design a new COMSOL experiment to measure the force between them when they are charged with equal *but opposite* voltages. Make a graph of force vs. separation between the plates. You'll need to make appropriate simplifications and choose experimental parameters on your own.

**Question:** In what ways is the force between parallel plates qualitatively different than the force between spheres?