LAB 0: Orientation to Remote Login and Multisim Live

**Disclaimer: Any saved documents are not permanent on VLSI lab computers
unless saved on the H: drive.
Documents saved on drives other than H: will be erased.**

**It is also suggested you save a backup on a networked platform
 such as OneDrive or Google Drive**

Purpose:

This semester of ELEC 3105 is being delivered completely online, which includes the laboratory component. As you progress through the course you will be required to complete versions of the ELEC 3105 labs that have been modified in a way that they can be completed at your place of residence as long as you have a reliable internet connection. This document is an introductory laboratory designed to get everyone familiar with the tools you will need throughout the term to complete the assigned lab component.

This lab is worth 1% of your final grade and it is important to complete the exercises in this document to ensure you have the base skill set needed to complete the core labs in an efficient and effective manner. For more information about the expectations of each student regarding the labs please refer to the course syllabus posted on cuLearn.

Resources Needed – *these will be needed throughout the term*

(i)   Device with Windows/Mac/etc. operating system (desktop or laptop is highly recommended)
(ii)   Reliable internet connection
(iii)   Carleton email address and password

Part 1: Connecting to Carleton's VPN (Virtual Private Network)

A large portion of the laboratory component for this course will require you to have remote access to equipment and devices on campus. To ensure the security of the university networks, all students registered in this course will be required to use Carleton's VPN service for remote login purposes. To set this up go to this URL:

https://carleton.ca/its/help-centre/remote-access/

Click on the instructions corresponding to your chosen operating system as shown in Figure 1.
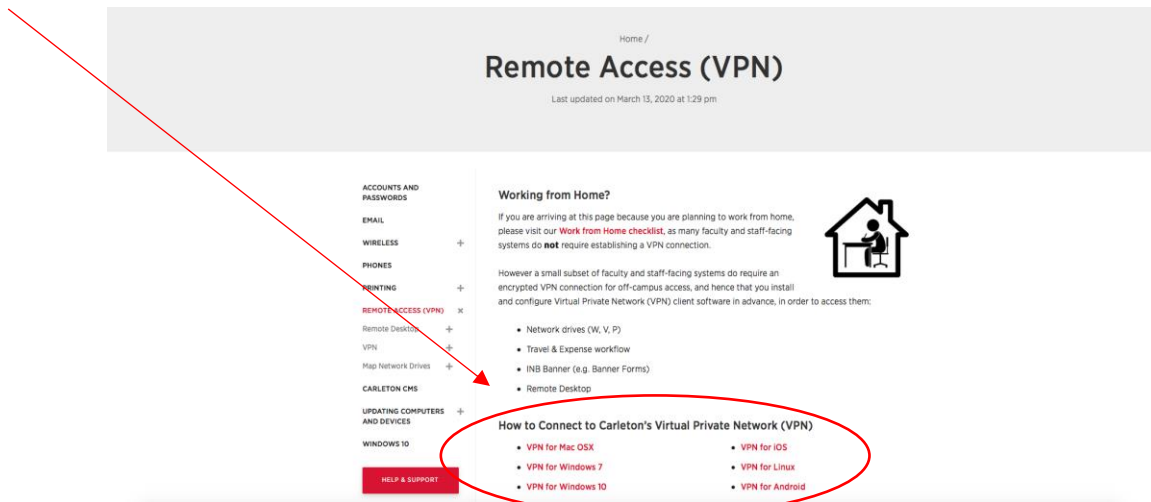
Figure 1: Carleton Remote Access (VPN) Website

You will be taken to a page providing instructions for you to set up your VPN based on your operating system. Once set up, you must connect to Carleton's VPN (**cuvpn.carleton.ca**) before accessing the lab computers and required equipment (when necessary).
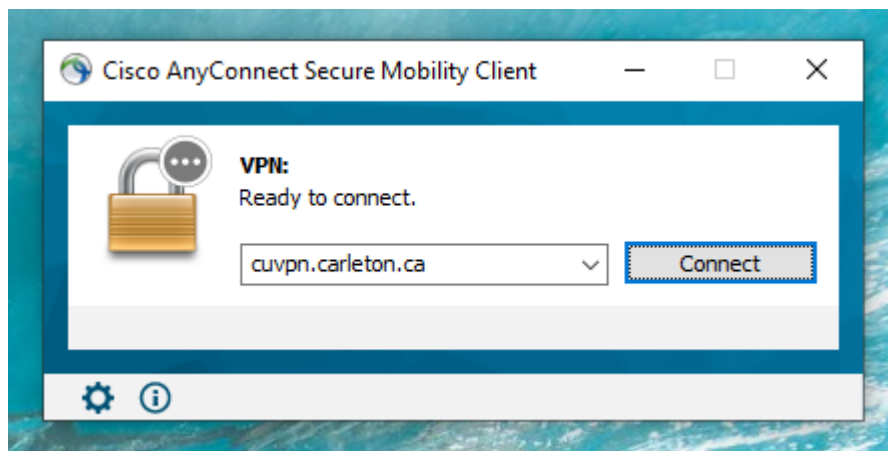

Figure 2: Carleton VPN Connection Setup

You will be prompted for your **MyCarletonOne (Carleton Central) username and password** to log into the VPN. Note that this is different than the login information used to access the lab server and computers. An example of logging into the VPN using Windows 10 can be seen below in Figure 3.
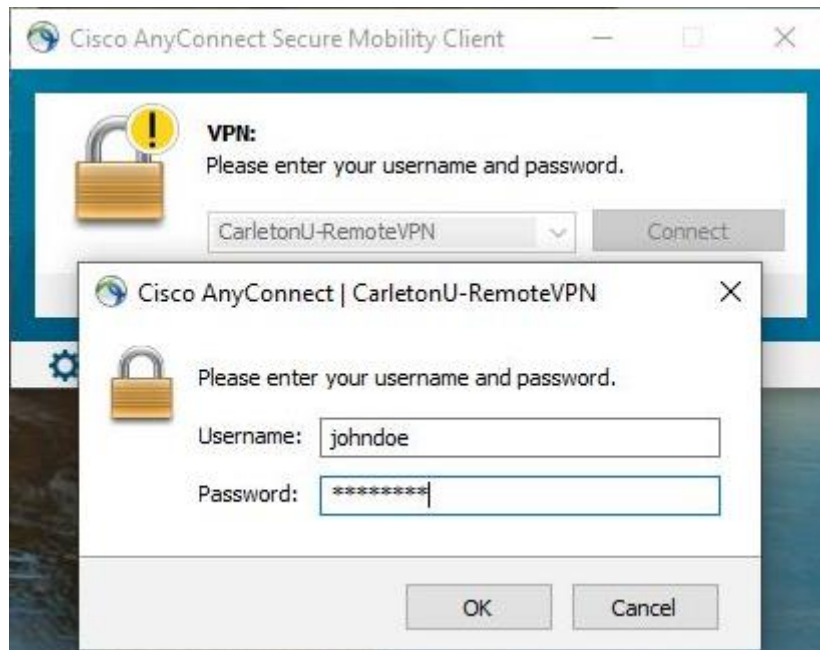
Figure 3: Cisco AnyConnect on Windows Computer Accessing Carleton VPN

If you have any issues setting up a VPN connection, please contact Information Technology Services (ITS) for assistance. Their contact information can be found in the link below.

https://carleton.ca/its/contact/

Part 2: Remote Login into VLSI Lab Computers

Once you are connected to Carleton's VPN you can access the lab server and computers. Logging into these computers also gives access to your H: drive, where you should be saving your files. When working on lab computers, do not save files to locations other than your H: drive. This includes the Desktop, Documents, Downloads, and C: drive. These files not only slow down the computer but will also be periodically deleted.

First, you must login to the VLSI server using a remote desktop connection. Then, from the server you will use another, nested, remote desktop connection to connect to the VLSI lab computers. From the lab computer desktop, you will be able to access ANSYS and/or control the lab equipment (if necessary). Figure 4 below shows this process.
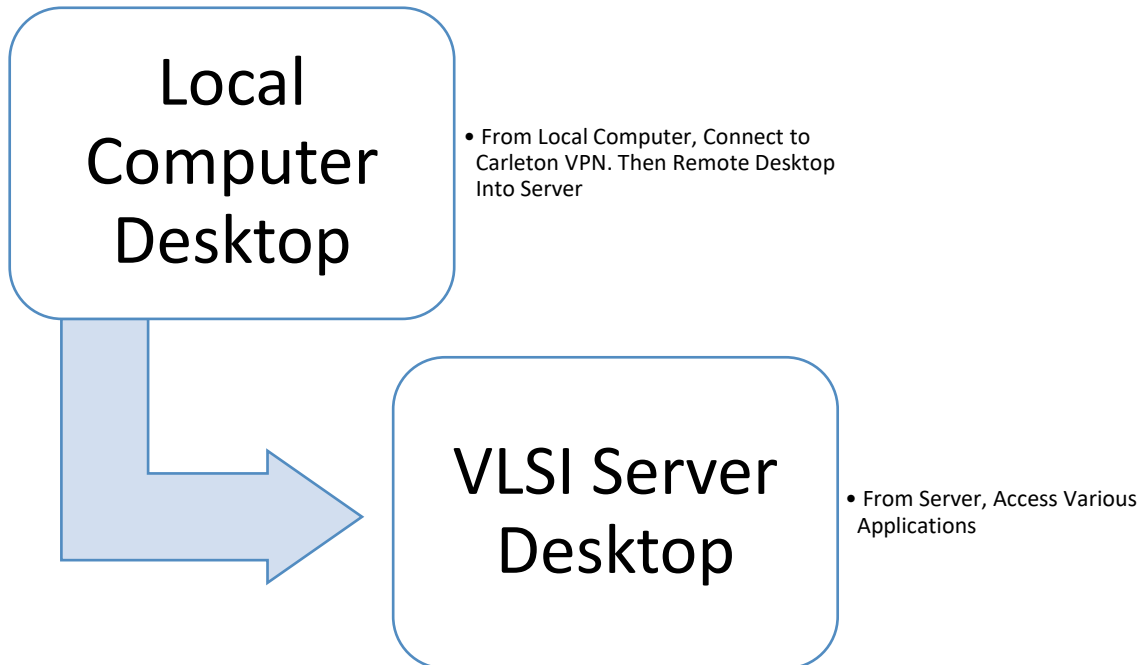
Local Computer Desktop

• From Local Computer, Connect to Carleton VPN. Then Remote Desktop Into Server

VLSI Server Desktop

• From Server, Access Various Applications

Figure 4: Remote Desktop Hierarchy Diagram

To login to the VLSI server, open the Remote Desktop Connection app if you are using a Windows computer, or the appropriate equivalent on another operating system. The shortcut icon for this app can be seen in Figure 5.

Figure 5: Windows Remote Desktop Connection Shortcut Icon

Once open you will be greeted by a window like the one in Figure 6.

Figure 6: Windows Remote Desktop Connection Window

At the bottom right, click **Show Options** to expand the **Computer** and **Username** tabs.

For **Computer** enter 134.117.38.204 (Eli), and for **Username** enter VLSI**{cuLearn Username}**. For example, if your cuLearn username is 'johndoe', the Username would be 'VLSI\johndoe'. This can be seen in Figure 7 below.
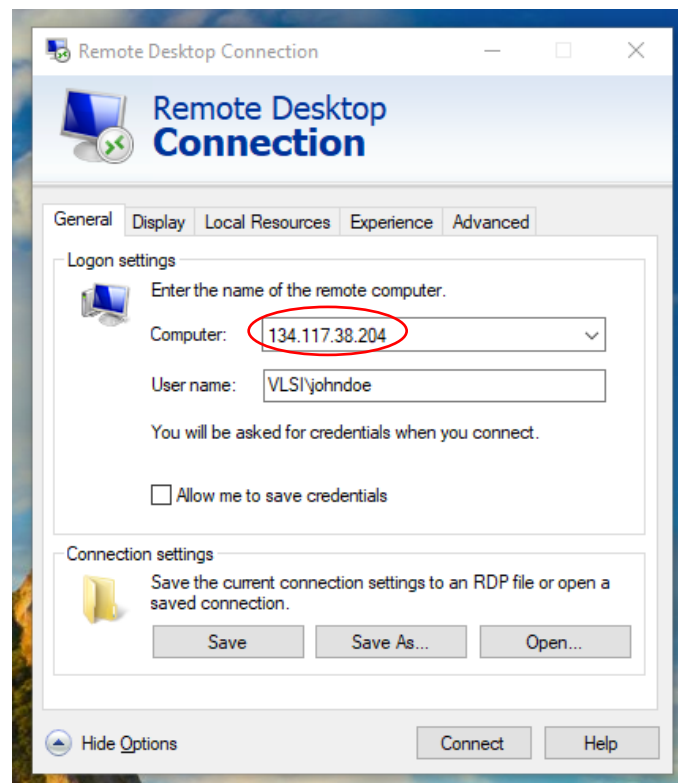

Figure 7: Expanded Remote Desktop Connection Window

Click **Connect**. You will be prompted for a password. This password is pW{**Student Number**}. So, if your student number is 123456789, then your password is 'pW123456789'.

This will bring you to the VLSI server. From here you can access the licensed software for completing your work during your labs. You can also organize your remote desktop into compartments using the Fences application. This can be seen in Figure 8.

Figure 8: Remote Desktop App Connecting to PC## From Server Virtual Desktop

Note that the remote desktop connection acts as a window, and can therefore be minimized, resized, and closed. This can be seen in Figure 9. If closed, your remote access session will disconnect, but it will leave you logged into the computer/server without closing the programs you were using. This can lead to major issues for any users after you, including not being able to control the equipment, or losing saved files. **Always log off the Remote Computer after you are done your work**.

When you are done with the remote desktop session, **make sure you log off the lab computer**. This will also end your remote access session. Do this by selecting the Windows button in the bottom left-hand corner and selecting logout. This can be seen in Figure 7. Trying to use CTRL + ALT + DELETE will bring up the options for your local computer, not the remote session.

Part 3: Getting the ANSYS Example Working

Ansys Maxwell will be used for Lab 1 and Lab 2 for performing electromagnetic simulations. To perform these simulations, you will need to be connected to the Eli server as shown in Part 2. You can fine Ansys Electronics Desktop on the server desktop as seen in Figure 9.
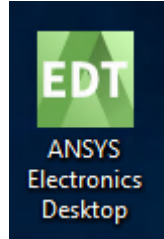

Figure 9: ANSYS Shortcut

Upon first starting up ANSYS, you will be prompted to change to a new ANSYS product library set, so click OK. In the Directory Options window which will follow, change the **Project Directory address to H:\ELEC3105**, if no such directory exists, make a folder on your H Drive for this. Override the default Temporary Directory and enter **H:\ELEC3105\temp\Ansys**, see Figure 10.
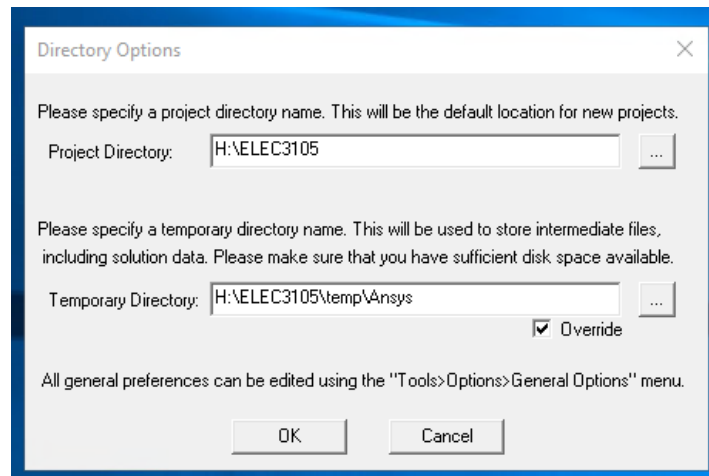

Figure 10: Directory Options for First Startup

Click OK, then Yes, No I am not willing to participate, OK, and Close. Once the application has open, you must choose **Project → Insert Maxwell 2D Design**.

With the new subproject open, you will see the X, Y, and Z normal vectors, and the scale legend. Open the example file by going to **File → Open Examples… → Maxwell 2D → Getting_Started** and open the Solenoid file. Below is the same Solenoid in 3D
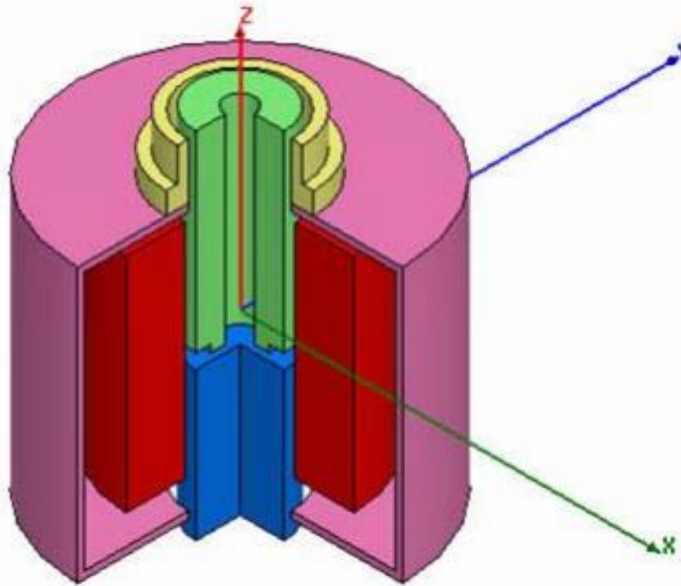
Figure 11: Open View 3D Equivalent of the 2D Solenoid Model

Since this is already constructed with a current flowing through the core (red), you can skip to analysing the schematic and displaying the magnetic field strength.

In the Project Manager window, the Solenoid project is displayed. You must expand the list and find Analysis, which can be expand to reveal Setup1. In Analysis, Setup1 is ready for analyses, right click it and Analyze. You may be prompted to save your project at this point if you have not done so. Navigate to the folder you created **H:\ELEC3105** and create a new folder for your project **lab 0.**

Highlight the whole bound space (click anywhere on the cartesian plane then Ctrl-a), right click **Field Overlays** from the project panel and choose **Fields → H → Mag_H → Done**.

At this point, the magnetic field strength should become visible. If it is not, expand Field Overlays and select the Mag_H in H. Other fields can also be selected for using the same method.

If you get lost at any point, or would like to perform this example from scratch, please follow the instructions seen here:
> https://www.physicsforums.com/attachments/tutorial-2d-magnetostatic-solenoid-pdf.78651/


Part 4: Running ANSYS Maxwell 2D

For Lab 1 you will need to make your geometries from scratch. Now that you have had experience loading a premade example in ANSYS, we will move through a few exercises that will help you gain familiarity with creating ANSYS geometries. ANSYS can be a bit frustrating a first, so it is important to get familiar with the order of operations such that Lab 1 can be completed efficiently.

> *Note: It is always a good idea to regularly save your projects (**H:\ Drive**) to prevent losing progress. If the instructions below are not clear for you, ask your TA or research online. Assume a plate thickness of 1mm unless otherwise stated.*

1.   Start the **ANSYS Electronics Desktop** program and select **Project**, then **Insert Maxwell 2D Design**.
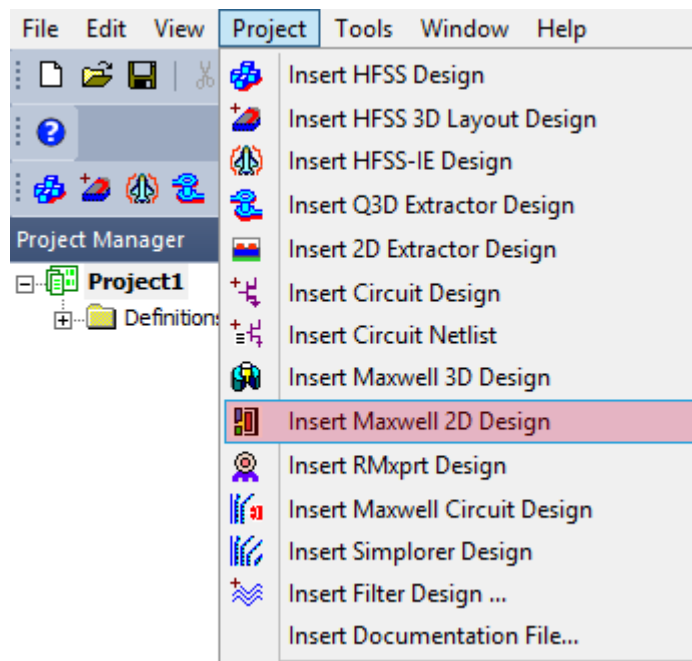
Now, select the **Maxwell 2D** menu option and click on **Solution type**.
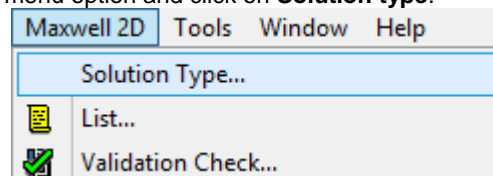
In the window the opens up, **select the required solution type**, for **lab #1 it is Electrostatic**, for **lab #2 it is Magnetostatic**. **Click OK** once you selected the correct option for you.

2. Click on the **Draw line** button shown below in order to draw the required structure geometry, depending on the problem you are currently trying to simulate.
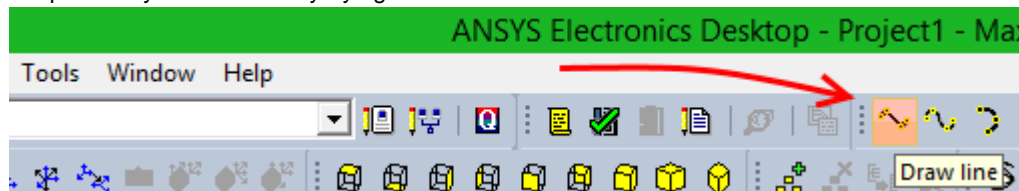
Click somewhere on the **white grid** located across the center of the interface in order to place a point, **click again** to place another point and a line will be automatically drawn between them. Play around with this function until you are comfortable using it to draw different shapes.

It is also possible to use the **rectangle** and **circle** functions in order to draw shapes. Familiarize yourself with how these functions work as well. You can zoom in and out by **holding the control key and scrolling** with your mouse, this may help in the case where you need a finer grid size. (Zoom in for a finer grid spacing).
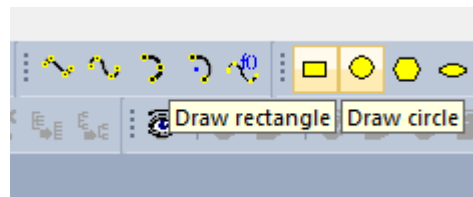
If a ring geometry is required, **draw one circle** with a radius equal to the radius of the ring's outside edge. Then **draw another circle** inside the first circle with a radius equal to the ring's inner edge as shown below.
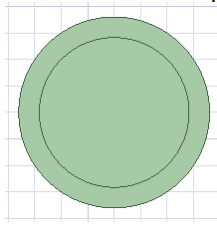

Next select both circles. You may do this by **holding the control key**, selecting the larger circle, then select the smaller inner circle, and **releasing the control key**. Once this is done, **right click** on the smaller inner circle, then navigate and click on the menu option shown below.
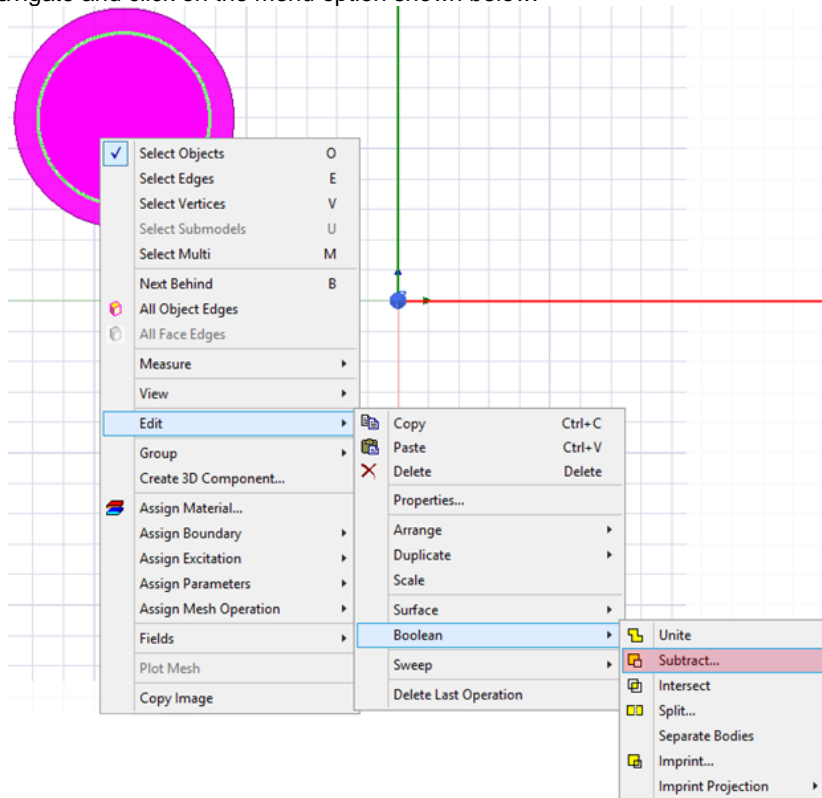
Click the **OK button** on the window the opens. You should be left with a ring as shown below (this may take a couple tries to get the swing of).
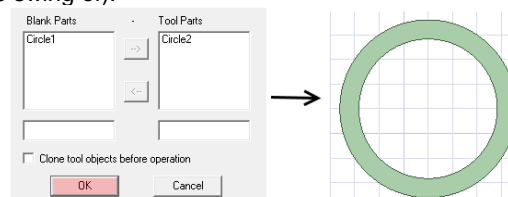
3. Once the required shapes are drawn according to the problem, **click on a shape** in order to select it. Once it is selected it will change color and its properties will be displayed in the properties pane on the left side of the interface. Select the property box containing the material value and **click on Edit** and a window will open. In the "Search by name" box, **type in the name of the material** you would like to simulate, **select it from the list below** the box, and **click OK**.
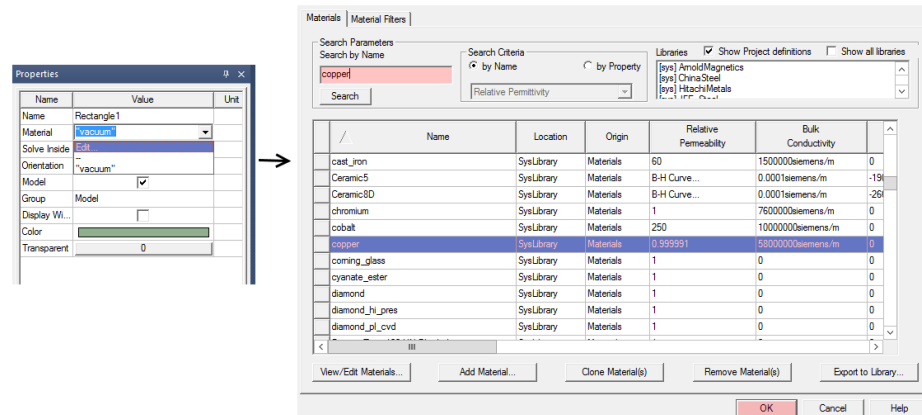
Do this for each different shape using the required material.

4. Once the geometries are drawn and have had materials assigned to them, **draw a large rectangle around them** and **set the material of this rectangle to air**. You may also change the transparency of this rectangle by changing the "Transparent" value located in the Properties pane.

    Once this rectangle is drawn, **press the E key[1]**, and then **while holding the control key**, **select all four edges** of the large rectangle you just drew that encapsulates the rest of the shapes. Now **right click anywhere on the white** grid and from the right click menu, **navigate to Assign Boundary (Balloon)**, and **click on Charge** (if you get a boundary error try carefully repeating this process). Set the Balloon type as **Charge** in the window that opens up, and **click OK**. You may now **press the O key** to return to object selection mode (as opposed to edge selection mode).

    It is also possible to hide or show the different geometries that you have drawn by **clicking on the eye icon** in the upper toolbar and selecting which shapes you want to remain visible in the window that opens. This is shown in the figure below. Note that if a shape is not visible, it will still be included within the simulation.
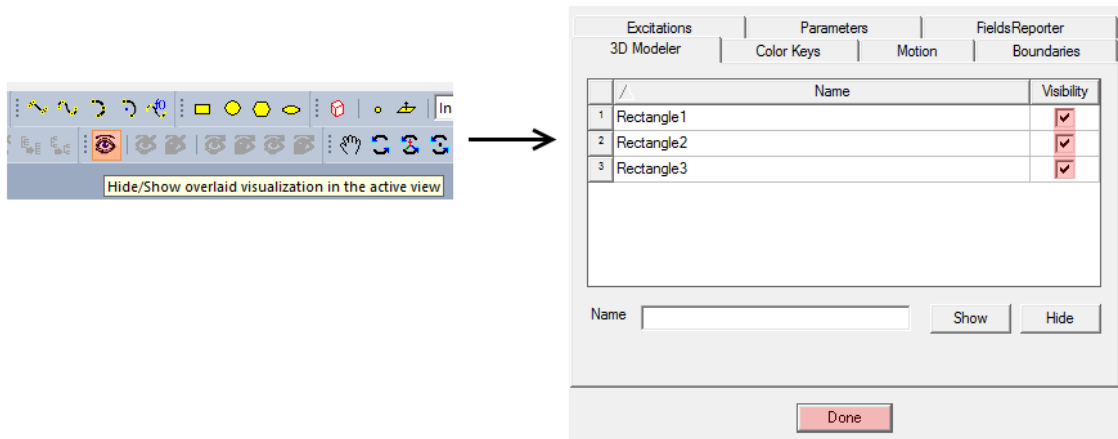

5. In order to assign voltages to materials, **right click** on the material that you wish to assign an excitation (for example a voltage), **navigate to Assign Excitation** in the right click menu, and **click on Voltage**. **Set the voltage value** required and **click ok**.

6. Verify that the shapes drawn have the correct size ratios. You should now change the scale by navigating to the top **Modeler** menu and clicking on it, then select **Units** and click it. On the window that appears

---

[1] The E key enables edge selection mode. Other common options are Objects (O) and Faces (F). Objects is the default selection mode.

check the **Rescale to new units** box and select **cm** or the proper unit for your drawing. Play around with this until your units are correct, you may verify if they are correct by using the scale located at the bottom of the white grid workspace interface shown below in the example for problem 1. (The copper plate thickness is set to 1mm along y in this figure.)
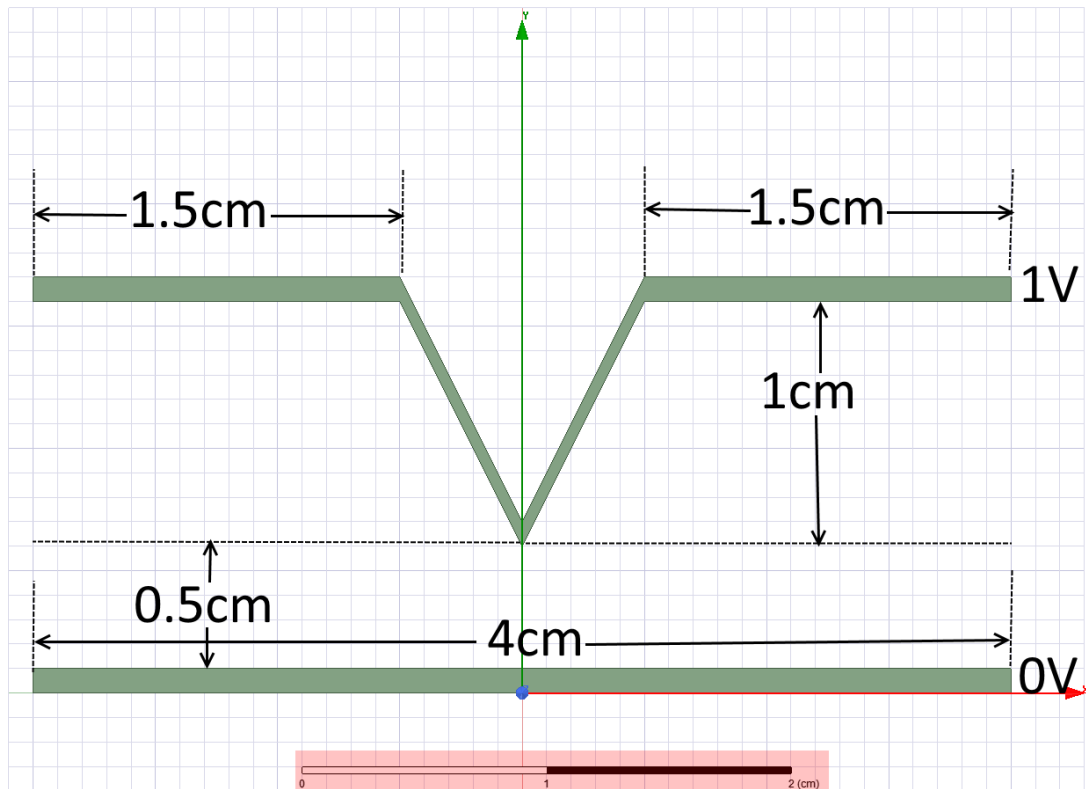

7. In order to simulate this project you must first add a solution setup by **navigating to the following menu shown below** and clicking on **Add Solution Setup**.
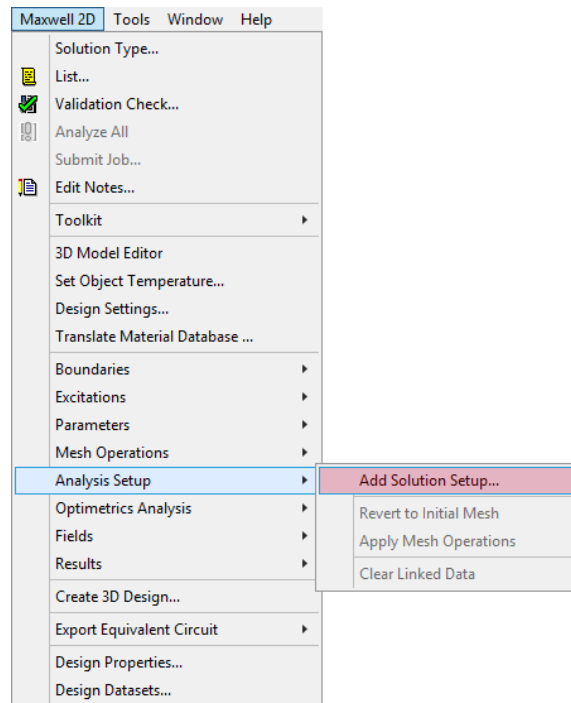
Click OK on the window that opens and then **save your project** by pressing down on the **control key and S key** at the same time. Choose a suitable location to save your project if required.

8. In order to simulate the project, navigate to the **Project** menu and click on **Analyze All**, as shown below. You must repeat this if you make any changes.
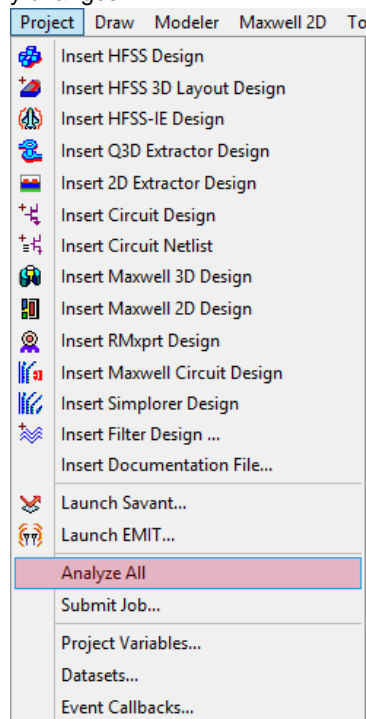
9. To plot the results, select the large air rectangle and right click on "**Field Overlays**" within the **Project Manager pane** on the left side of the interface as shown below. Select the type of result you would like to plot and a window will open. Click **Done** on the window and your plot will be visible.
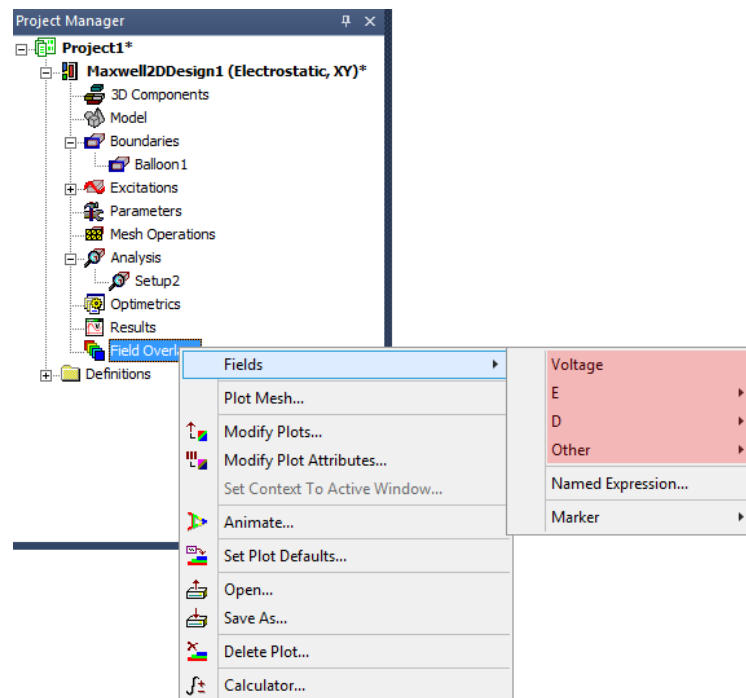
10. To find the capacitance, before analyzing the problem navigate to the project manager pane on the left side. Right click on **Parameters** and assign a matrix. Select one signal line and one signal ground (one for each source). Click OK then run the simulation. Once the simulation is completed, expand the **Parameters** tree, right click on **Matrix1** or the matrix you created, and choose **View Solution**. The capacitance units are shown near the top-right of the window and the value is shown in the large display box in the bottom half of the window.

11. Explore the **Project Manager** pane as it contains lots of useful information. You can modify field overlays by **right clicking on Field Overlays** and clicking on **Modify Plot Attributes**. This is useful for changing the resolution, color, and scale of the legend. You may also see your excitations and boundaries along with other parameters. Explore the **Properties** pane for useful settings too. You can rename shapes that you've created to custom names for easier identification if needed.

This brief tutorial on using the ANSYS Maxwell Solver should be enough to get you started, there is plenty of documentation available on the internet as well as built into the program itself.

Once you have completed 10., check in with your TA (Zoom or BBB) to let them know you are done. Your TA will mark you as present and this will be sufficient for the 1% credit for this lab.