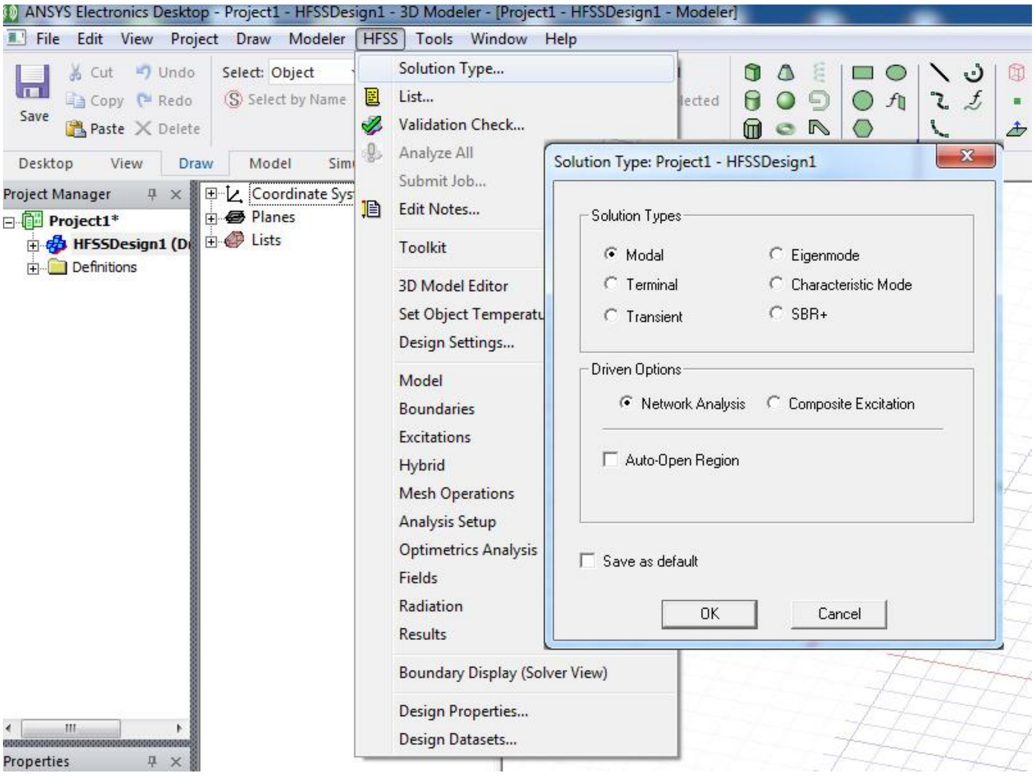
Laboratory 4

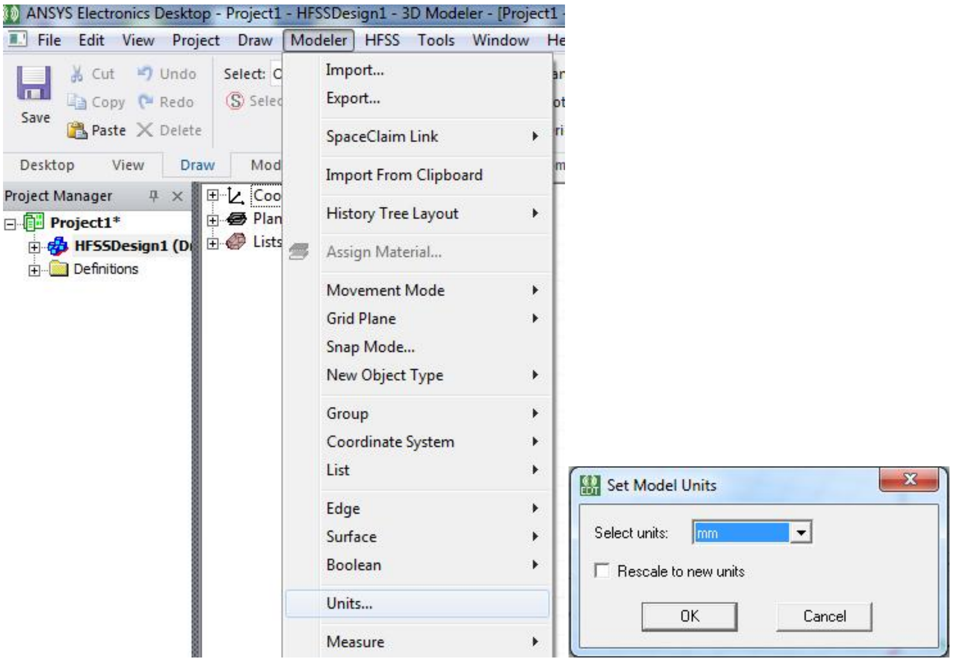
Simulation in HFSS, Dipole Antenna, part I

Start ANSYS Electronics Desktop, then start HFSS by clicking on the corresponding icon in the ribbon. If you forgot how, access laboratory 2.

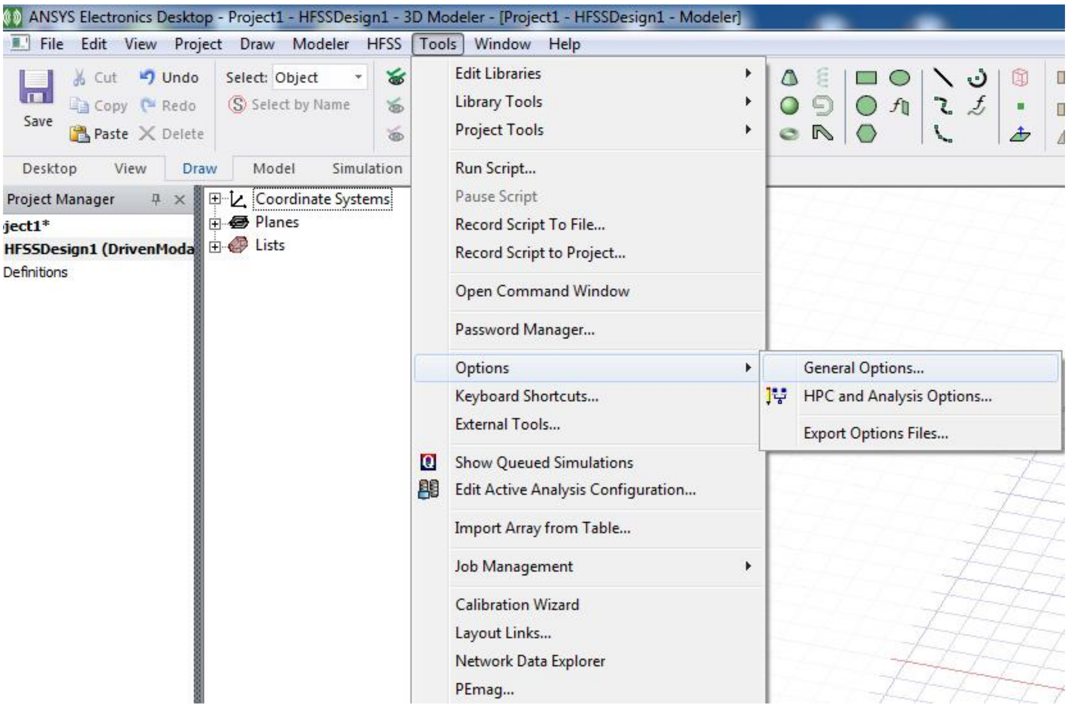
Check (and change if necessary) that the working mode is Driven Modal, **HFSS** > **Solution Type** > **Modal** .

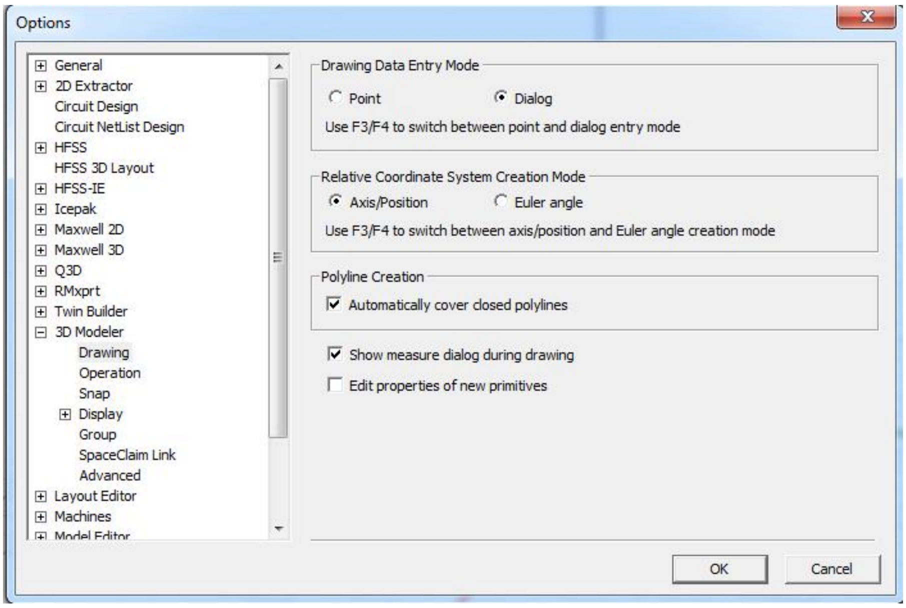


Set Modeler length units to **mm** , **Modeler** > **Units** > **mm** .

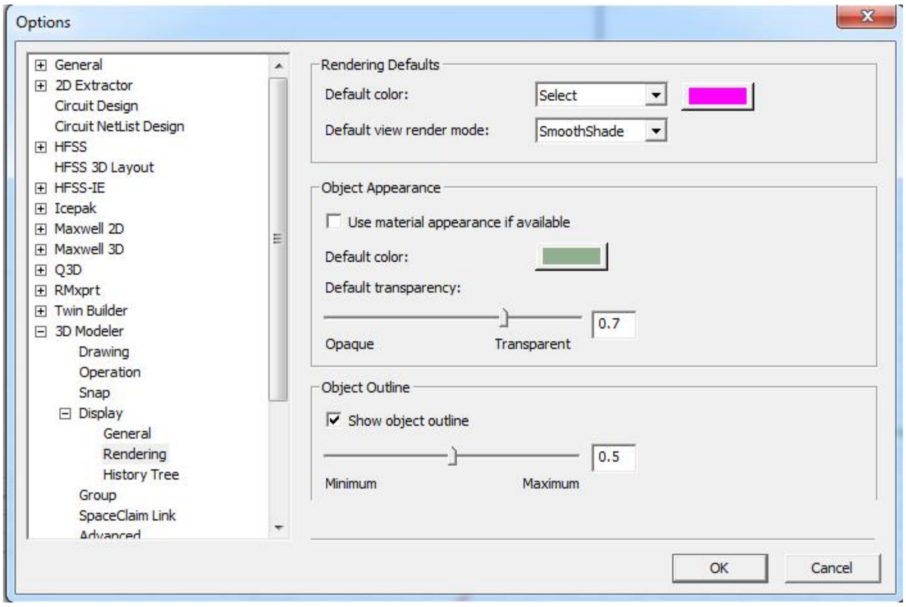


Change the default drawing mode to **Dialog** Data Entry mode (the coordinate input window), **Tools** > **Options** > **General Options** > **3D Modeler** > **Drawing** > **Dialog**. Remember that the F3/F4 keys let you switch between this mode and the **Point** mode (using the mouse).



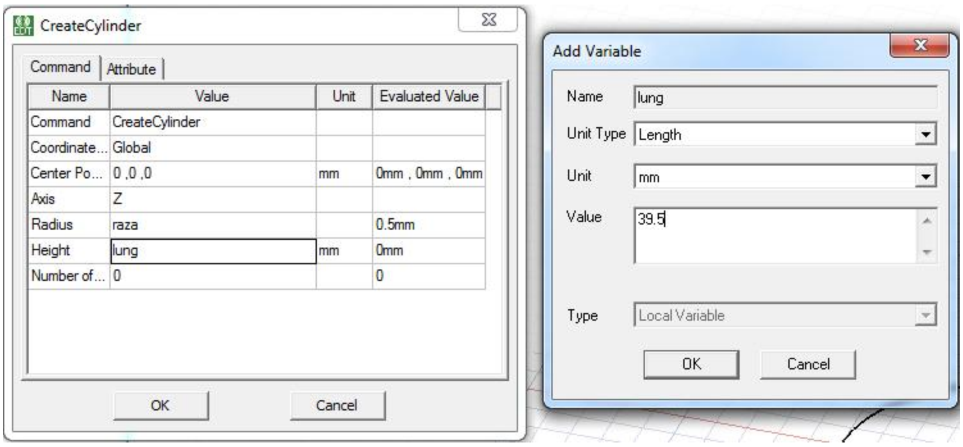


For a better visualization, set a default transparency of 0.7, **Tools** > **Options** > **General Options** > **Display** > **Rendering**.



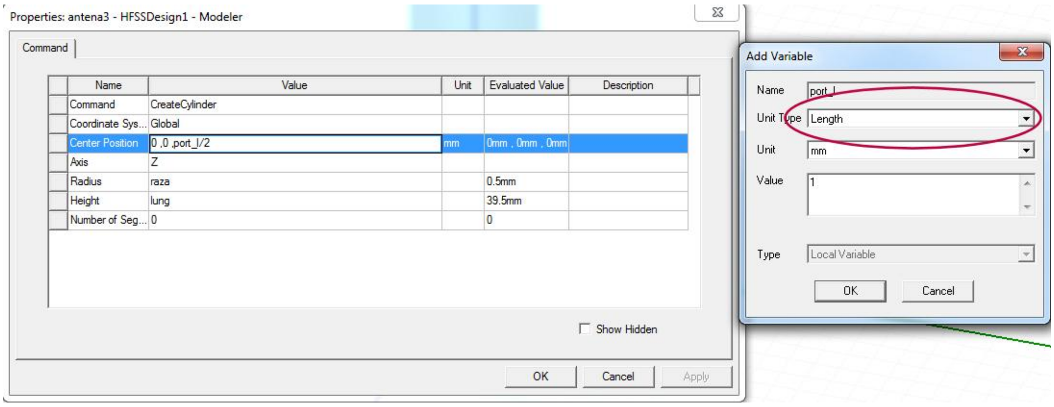
Draw the antenna in Parameterized mode. This mode assumes that variables will be used for the dimensions of the structure. For the dipole, three variables are needed, the radius and length of the two wires plus the distance between the wires. The parametric mode has the advantage that the model dimensions are entered by variables that receive an initial value, and, if necessary, changing the numerical value of the variable leads to the automatic redraw of the structure taking into account the new value. Optimetric (Parametric) analysis is also available in HFSS (automatic variation of a variable, with comparison of the results - for example we change the radius in steps and we check the effect on the resonant frequency of the antenna).

Variables are defined while drawing the model. If a word is entered in one of the input boxes that require a length, if that variable does not exist, the window for its definition and entering the its initial value appears. To draw a parameterized wire, select **Draw** > **Cylinder** and enter names at radius and height position.

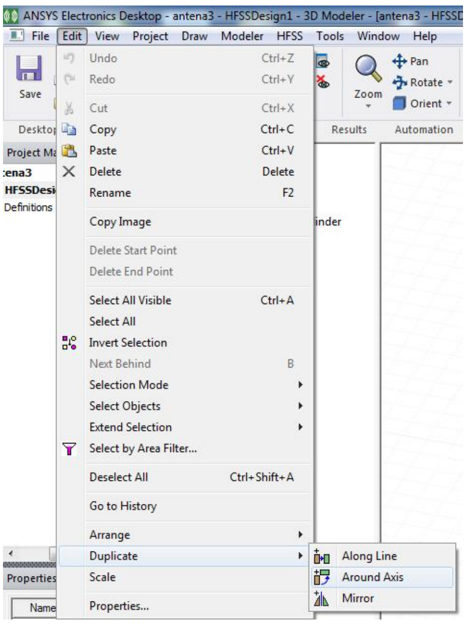


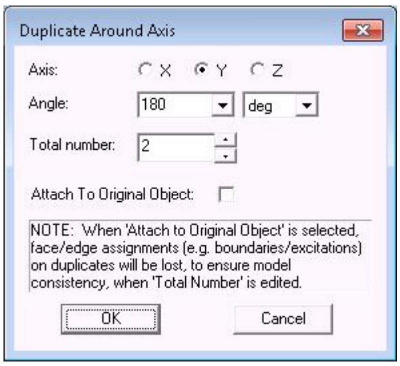
Check (and change if necessary) that the data type is Length with the unit of measurement mm. In the example, the names "raza", "lung", "port_l" are used as variables, with the initial values of 0.5mm, 39.5mm, 1mm respectively. Draw the antenna along the OZ axis, symmetrical with respect to the origin (hence the positioning of the center 0, 0, port_l/2).

Important! The names you use are not essential, you can use other words if you want, but it is essential to keep consistency throughout the lab (if you have chosen to name the variable "X" use the same "X" further where that value is used).



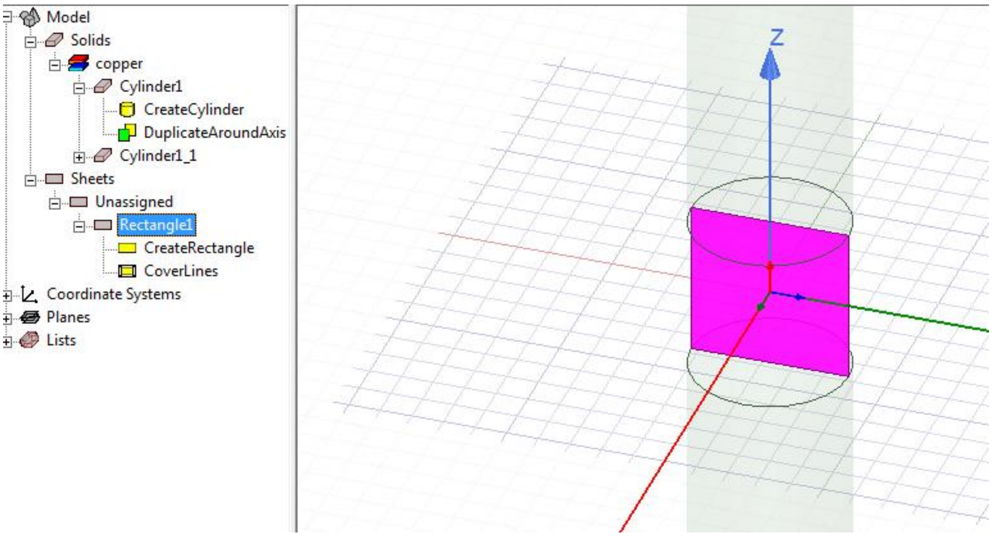
After drawing the first wire of the dipole we will create the second one by symmetry. Taking into account the particular way we chose the coordinates we can create the second cylinder by rotating the first one 180° around the OX or OY axis (! Not the default OZ). Select the first cylinder and then select **Edit** > **Duplicate** > **Around Axis**, choose the axis X or Y and enter the angle of 180°, Total number 2. The advantage of duplication is that the second cylinder will keep the same parameterized dimensions with the same variables names as the first cylinder, changing the variable value will redraw both.

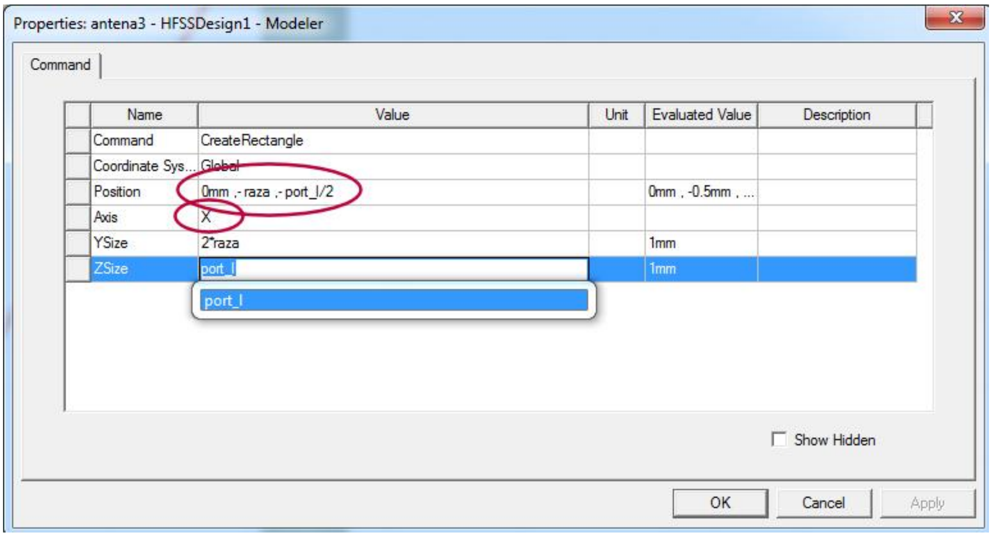




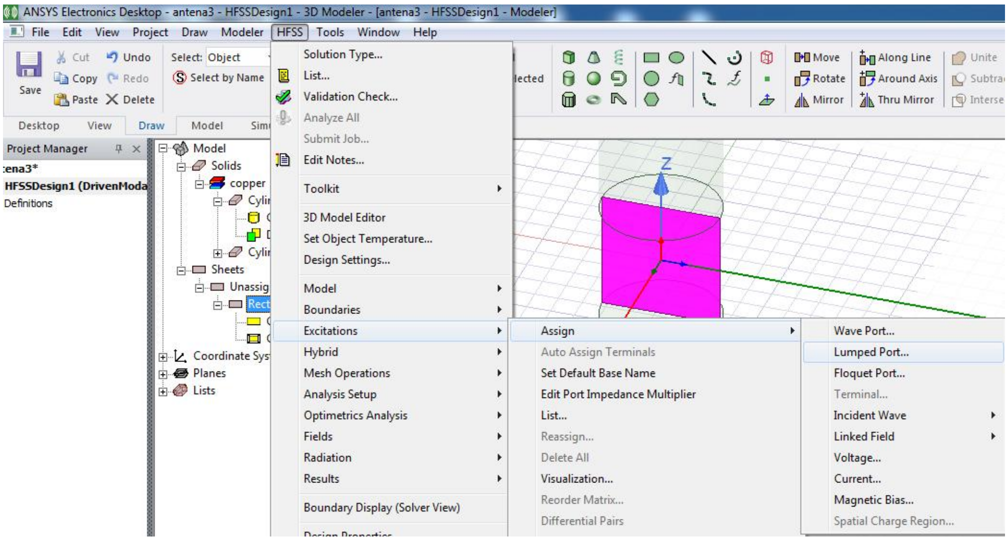
As in laboratory 3, assign a material (cooper) for the two wires.

We must provide a input source to perform the simulation. The input signal will consist in defining a reference electromagnetic field between the two wires. The area where the input signal is applied (the generic name for this area is **Port**) is drawn as a rectangle. Select **Draw > Rectangle** and draw a rectangle (! it's a surface model not a volume one) between the two wires, as in the figures bellow.

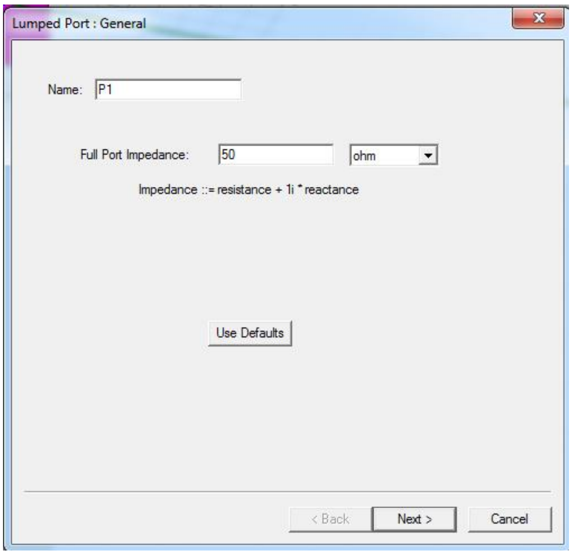


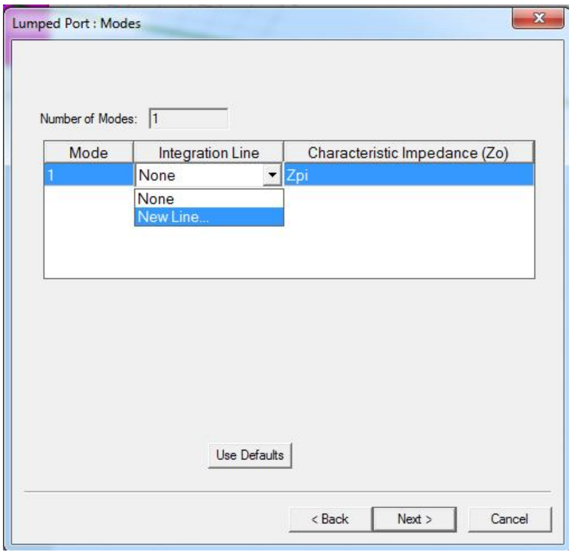


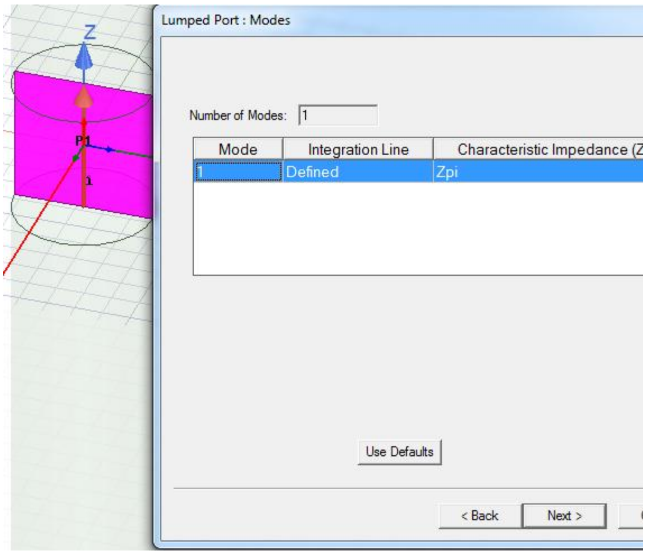
Select the drawn rectangle and define it as the input signal area/port **HFSS > Excitations > Assign > Lumped Port**.

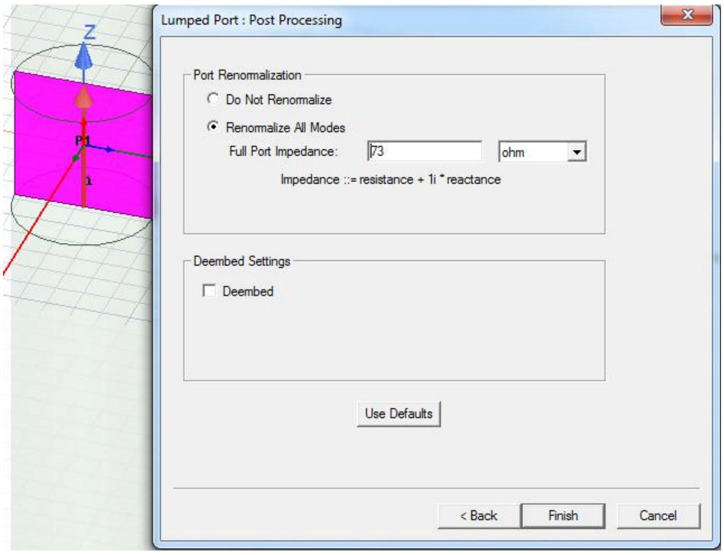


When defining the port, in three successive steps, choose the port name (P1), draw the integration line between the axes of the two cylinders, and enter 73Ω for renormalization impedance.

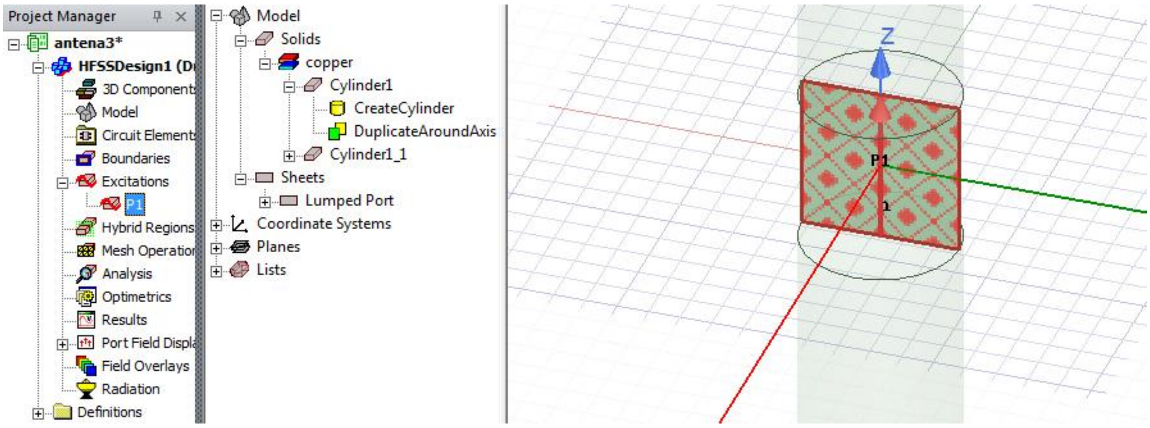




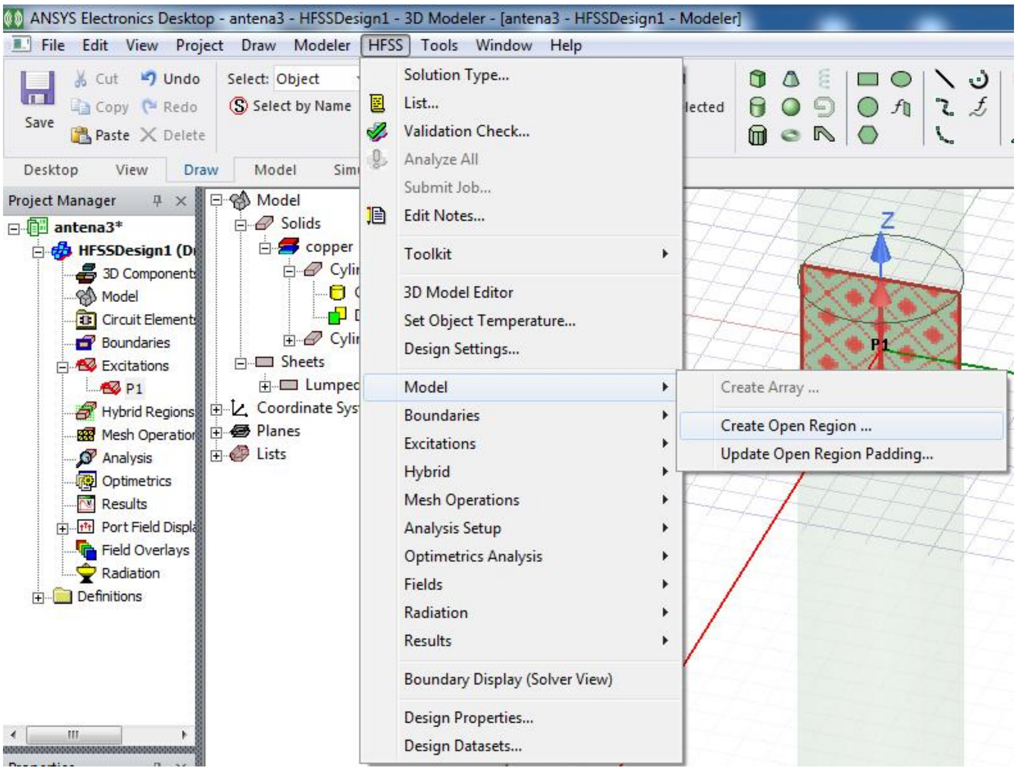


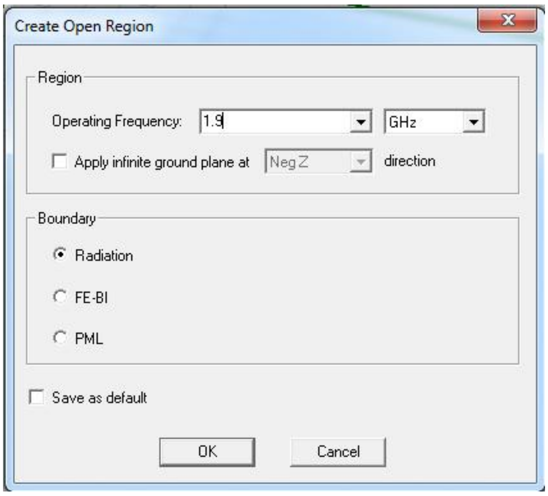


Check the success of port definition in **Project Manager > Excitations > P1**. The view on the screen should be identical to the figure bellow.

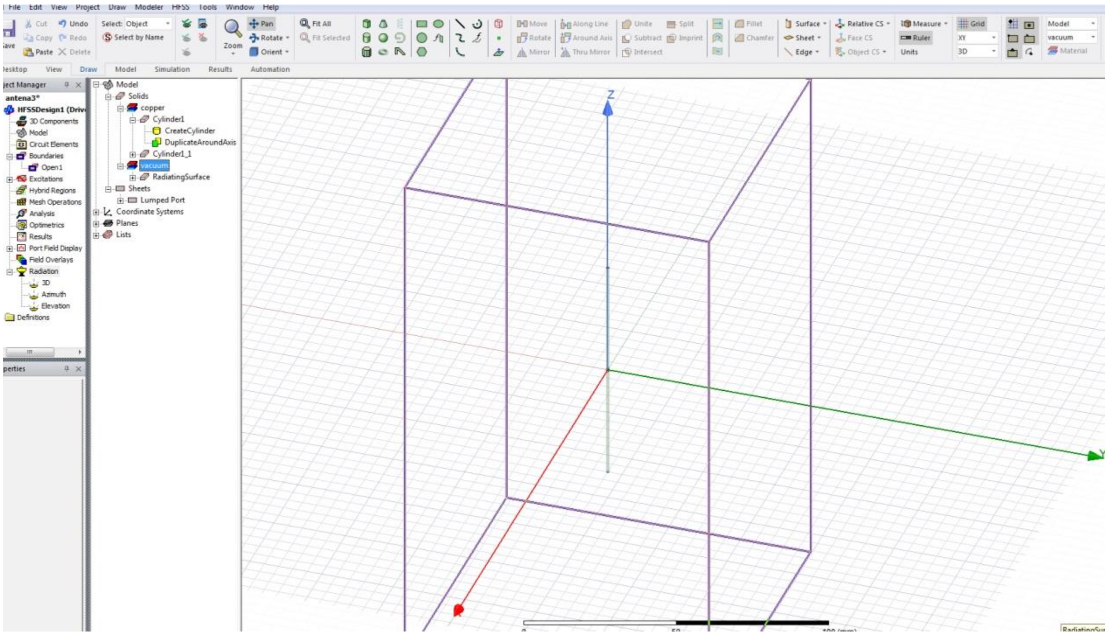


The computation area is defined ("Open Region": outside this area HFSS considers the simplified model of free propagation of the waves in vacuum, without influence from other objects, not even the antenna you drawn). "Open region" determines the transition from near/far field regions. Select **HFSS > Model > Create Open Region**. As in the figures bellow, choose the frequency of interest (around 1.9GHz) and Radiation boundary.

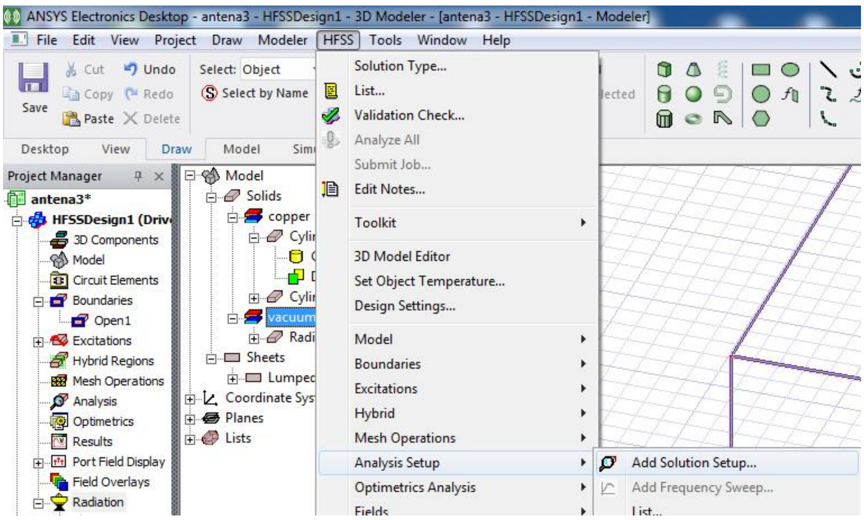




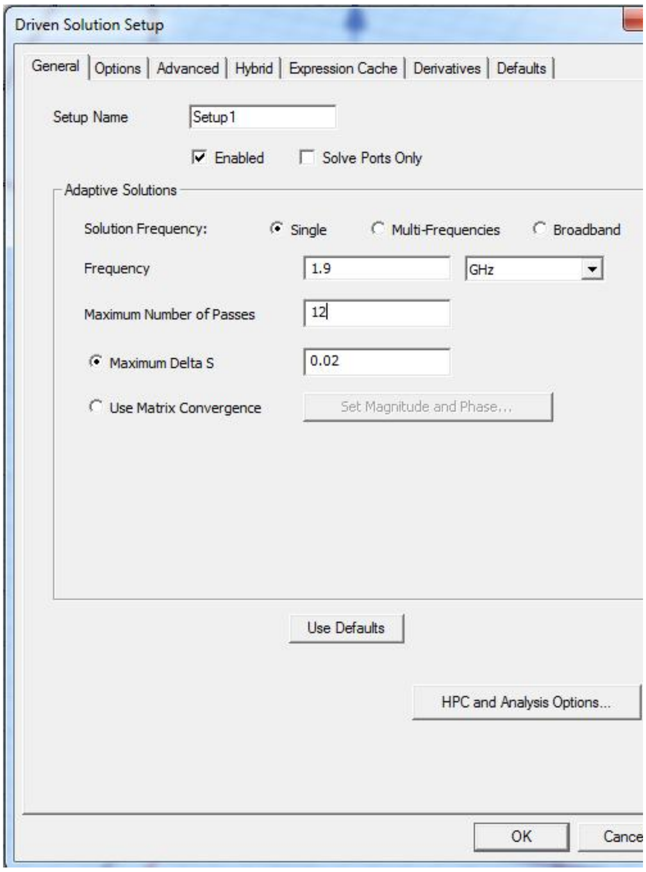
If successful, a rectangular radiation surface is obtained that will surround the two wires at the appropriate distance (for 1.9GHz near/far field regions).

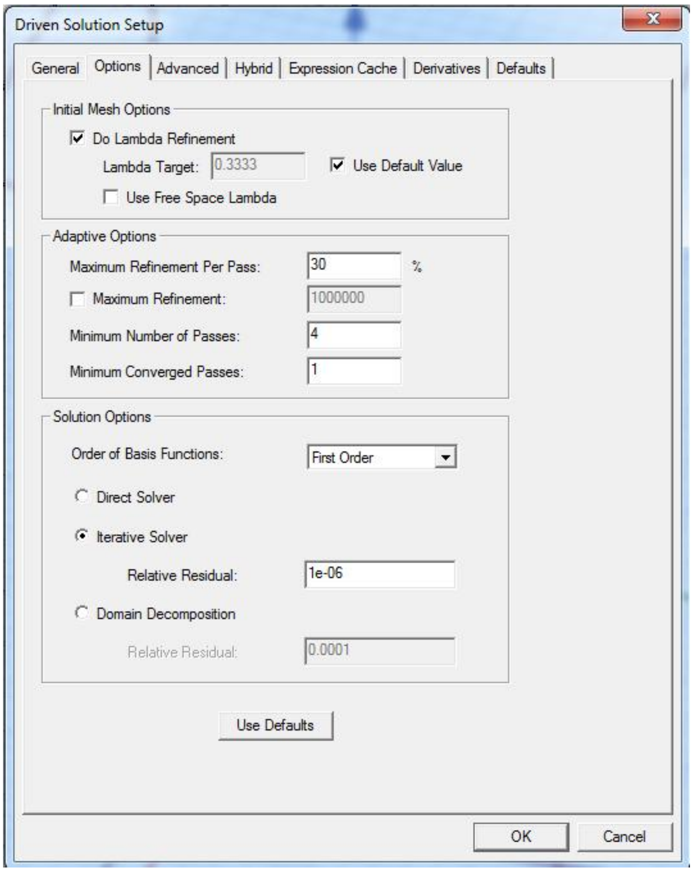


Add a solution. Select **HFSS > Analysis Setup > Add Solution Setup**.

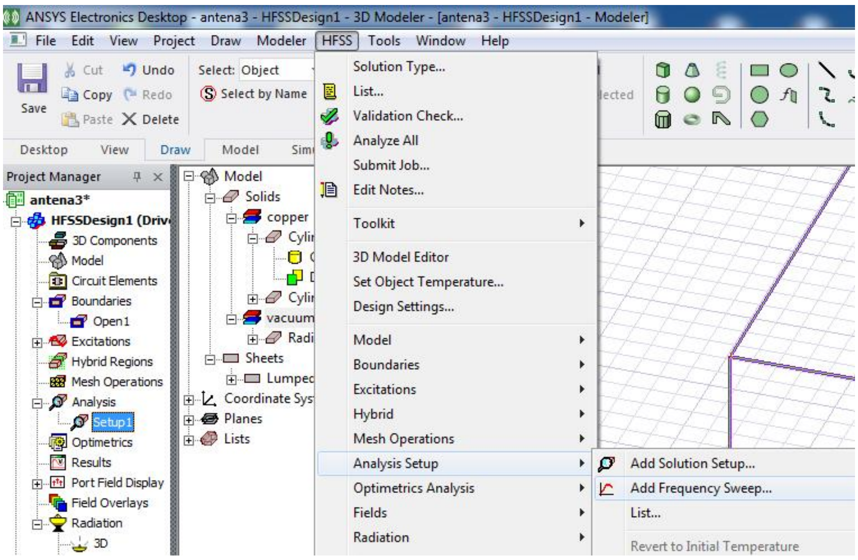


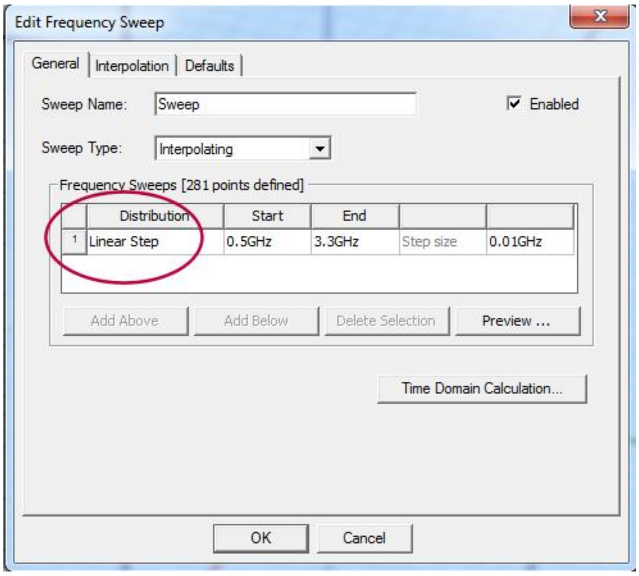
In the first two tabs of the window, choose the frequency around which you want to have the correct results (1.9GHz), the maximum number of passes 12, minimum number of passes 4 and select the iterative solver. The structure will be meshed at the indicated frequency.



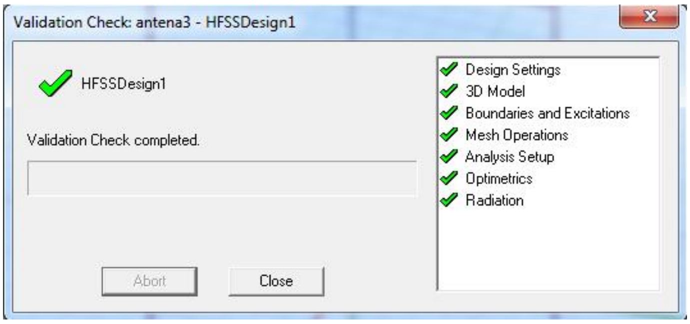


After meshing, we define a broadband analysis. Select the previously defined solution (**Project Manager** > **Analysis** > **Setup1**) and add a broadband solution around the frequency corresponding to the single frequency ssolution **HFSS** > **Analysis Setup** > **Add Frequency Sweep**.

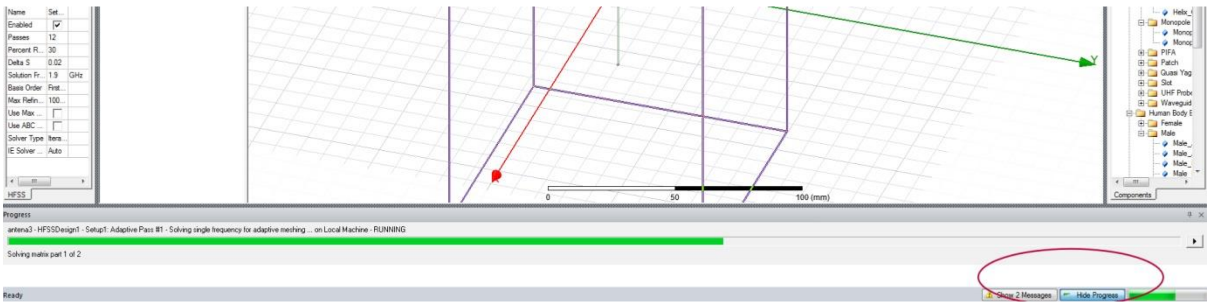




Select **HFSS > Validation Check** to verify that all required steps have been completed.

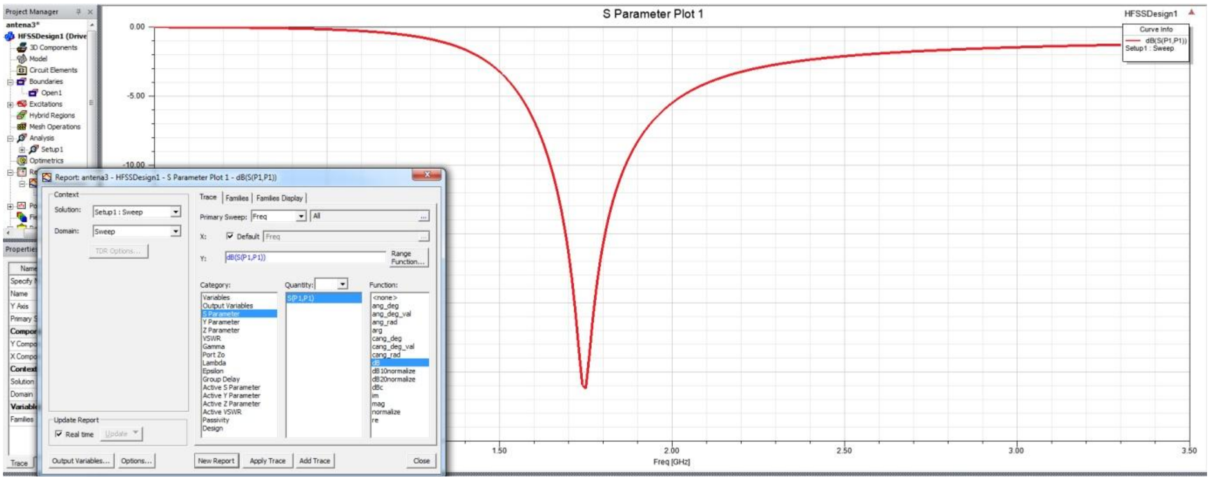


Select **HFSS > Analyze All** to start the simulation. Click the **Show Progress** button to view the progress of the solver.



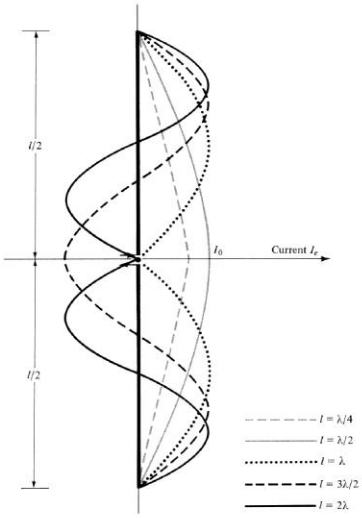
When the simulation is finished, we plot the results that provide useful information about the antenna.

Select **HFSS > Results > Create Modal Solution Data Report > Rectangular Plot** to display the S parameter corresponding to input port P1 (in dB). The resonant frequency of the antenna can be found. To find out the exact value, you can place a marker on the screen in the desired position, either by right-clicking on the graph and choosing the **Marker > Add Marker** command or from the menu **Report2D > Marker > Add Marker**.



Activity in the laboratory

1. Using the formula c / f = λ calculate the length of the wires required to obtain the resonant frequency corresponding to your own data.



2. Create the HFSS project according to the instructions in this lab with the dimensions corresponding to your assigned data (length and radius). Analyze the project. It is recommended to work with parameterized model (using variables) as it will significantly ease the work for the next laboratory. Most likely the resonant frequency will not be exactly the desired one, which is not a mistake. The antenna design activity will be continued in laboratory 5 to obtain the required frequency.

3. Save the project with your assigned data. Create an archive from the menu ("*.aedtz") with the simulation results included (set the "Results/solution files" option) and upload the archive to the lab server (it **must** be a **single** file).

Very important! If the archive size (including results) exceeds 10MB, it will not be possible to upload it directly to the server. In this case, use a cloud service (such as Google Drive or Microsoft OneDrive) where you upload the archive and submit to the server a download link to this file (check from another device/computer that the link allows download without username / password).

Reference

MARK JONES, Ansys High Frequency Structure Simulator (HFSS) Tutorial
https://indico.fnal.gov/event/13068/contribution/10/material/slides/1.pdf