# Build a co-planar waveguide in Sonnet

## Desired specs:
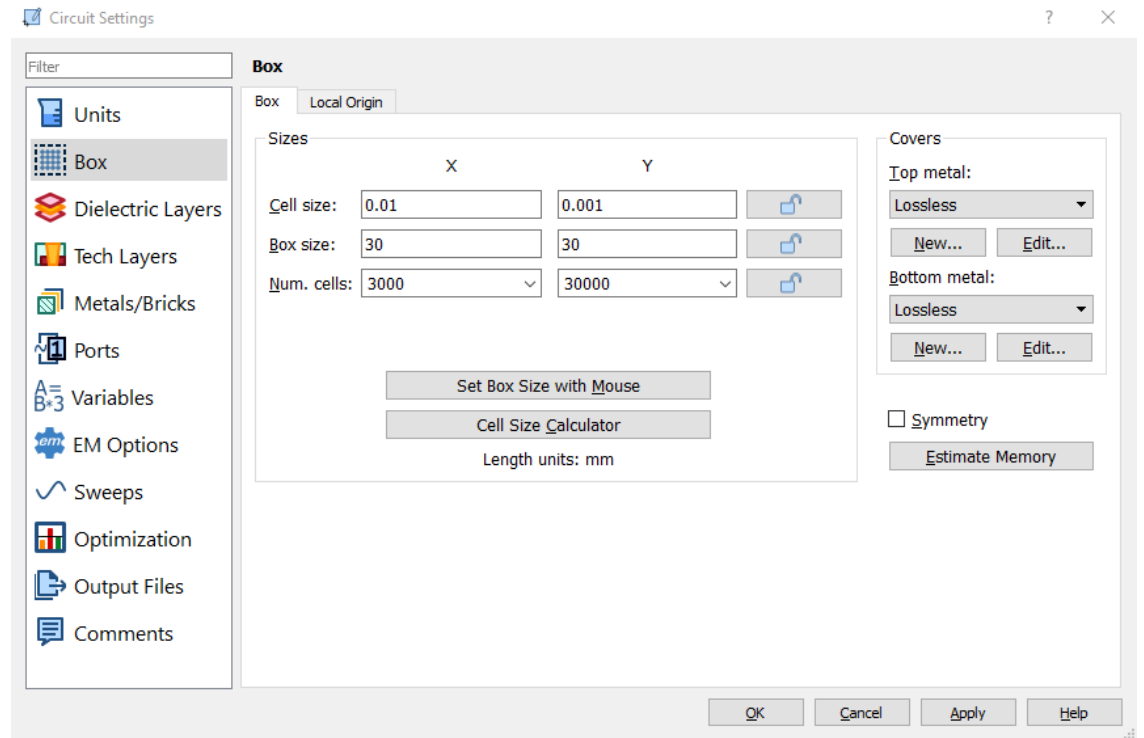
- 50 Ohm impedance as calculated by https://spok.ca/index.php/resources/tools/99-cpwcalc

- Aluminum Nitride substrate

- Grounded, with vias

## Setup
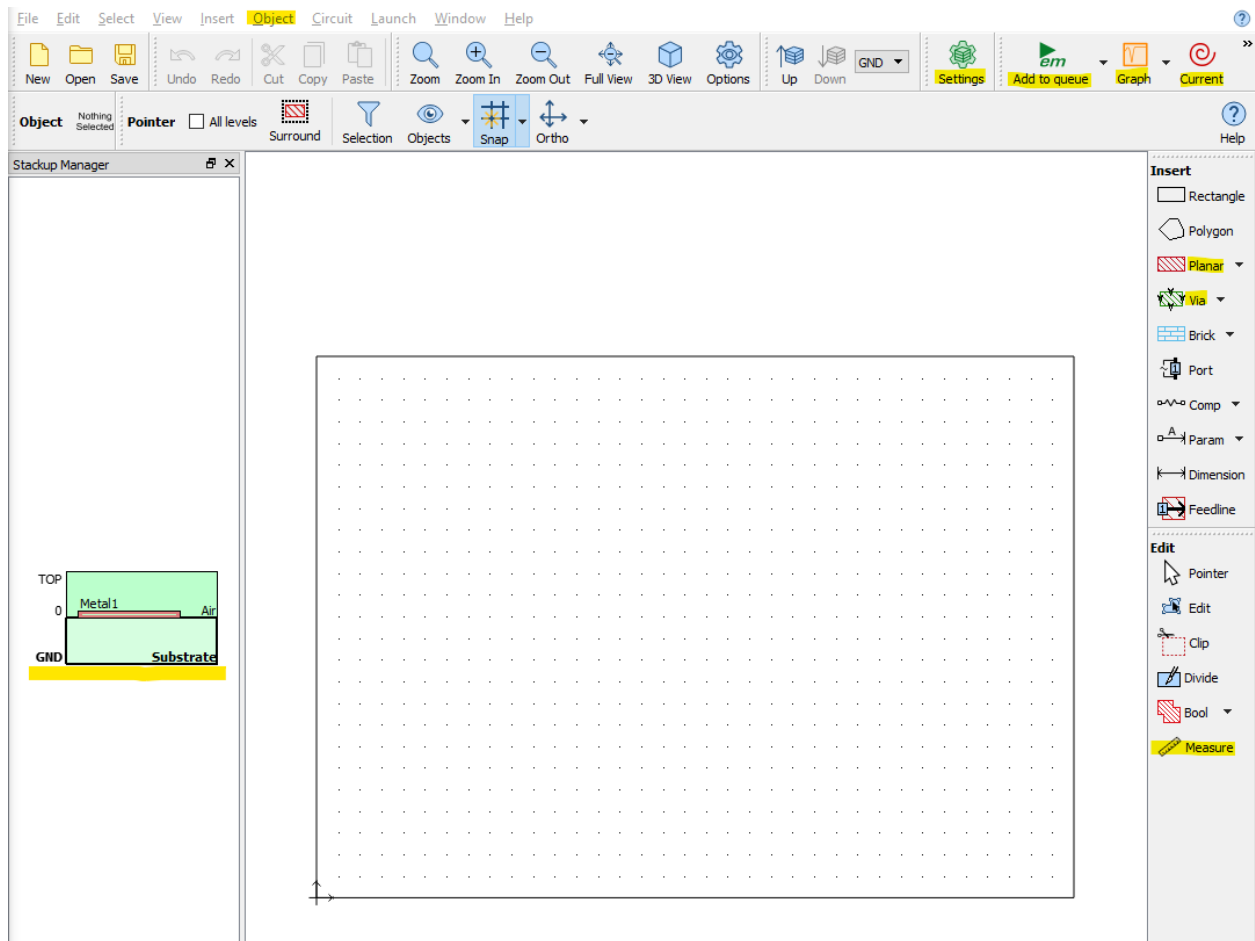
*Highlighted are the most important functions you'll need to use to build your CPW.*

1. Set up your box dimensions.

    a. Open `Settings` in the top toolbar.

    b. Under `Units` set `Length` to mm.

    c. Under `Box` set X and Y to match these dimensions:



    d. Under `Dielectric Layers` change the thicknesses of Level 0 (Air) to 200 mm.

2. Add your media.

    a. Set substrate to AlN:

        i. Double-click on the `GND` box in the left panel.

        ii. Open `Select Dielectric from Library`

iii.     Select `Aluminum Nitride` but do not click `OK`. Click `Edit` and type
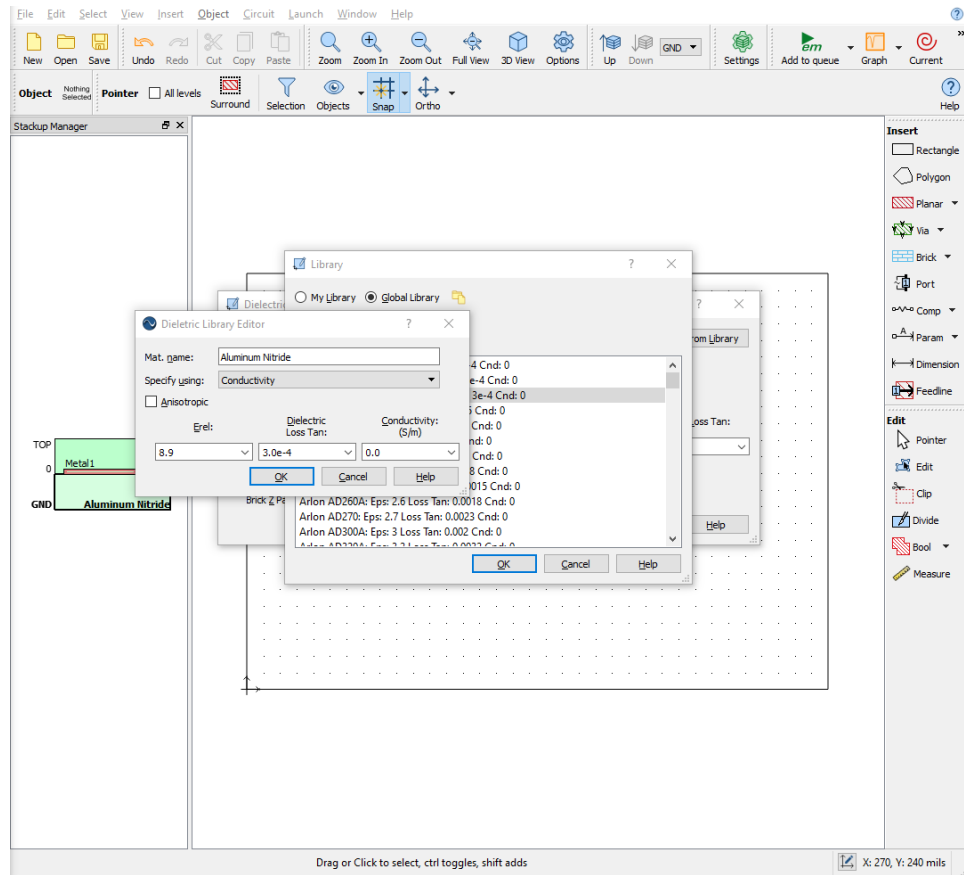


8.9 under Erel. Click `OK`.

iv.     Change the thickness of AlN to 1mm, and close all boxes.

b.  Add Copper for Trace.

i.      Double-click on `Metal1` inside the `TOP` box in the left panel.

ii.     Click `New…` to open the Metal Editor.

iii.    Click `Select from Library` to open the Library and select Copper.

iv.     Set `Thickness` to 0.005 mm.

v.      In the dropdown menu next to `Model`, select `Thick Metal Model.`

vi.     Change the name to Trace to differentiate from groundplane.

vii.    Click `OK`.

c. Add Copper for Groundplane.

    i.    Now right-click on the `GND` box and select `Add Metal Tech Layer`

    ii.    Repeat the steps from b., but do not change the model type to `Thick Metal Model`.
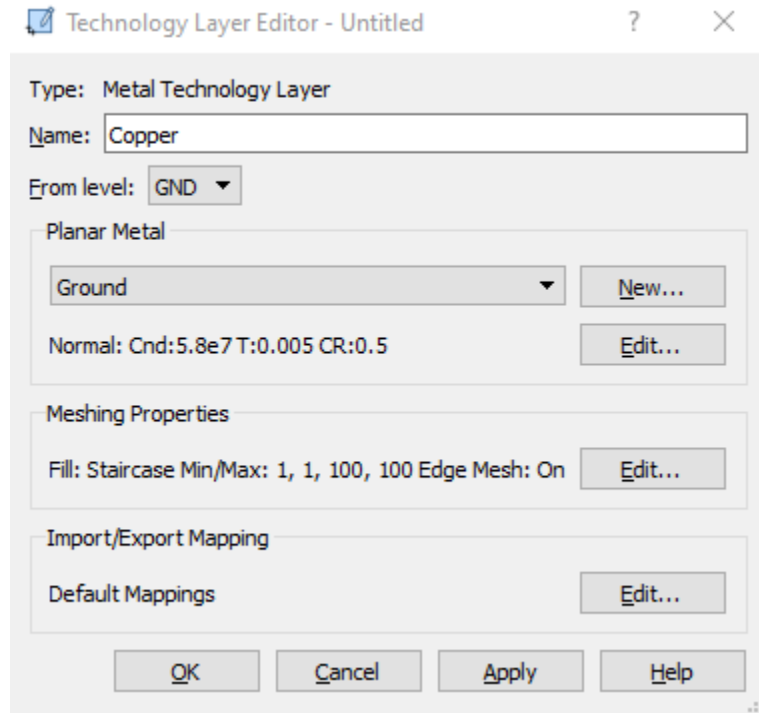


3. Put in your groundplane.

a. By alternately clicking between the GND and TOP layers in the left panel, you can toggle between the layers. Select GND.

b. On the righthand panel, open the dropdown menu for `Planar` and select `Draw Metal Rectangle.`

c. Fill the entire box size with this metal.

# CPW Components

1.  Construct the center trace and upper copper layers (UCLs).

    a.  Just like with the groundplane, select `Draw Metal Rectangle` from the dropdown menu beside `Planar.` Make sure you've selected the TOP layer.

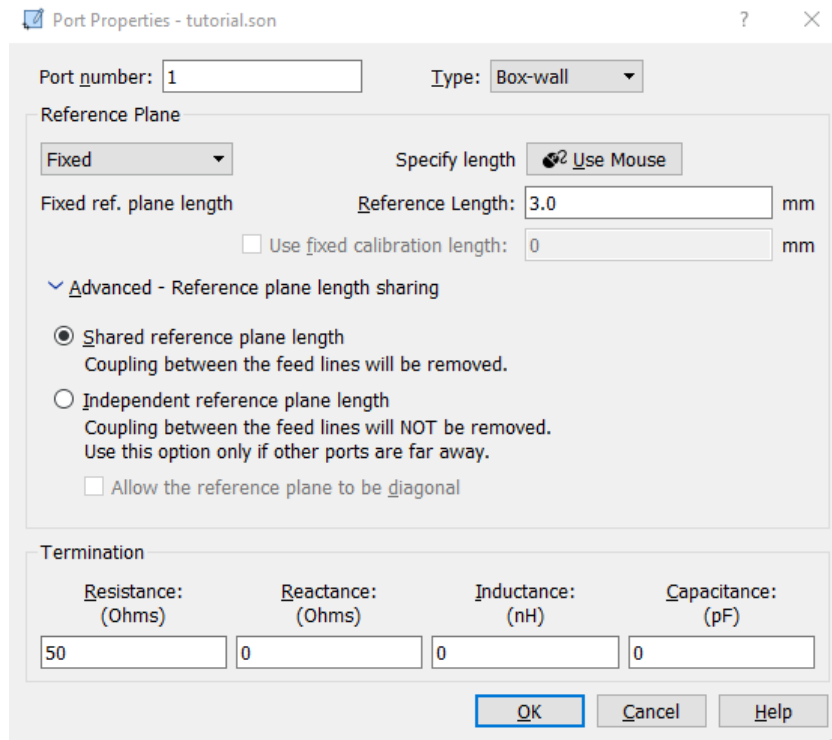    b.  Draw 3 rectangles across the box. The dimensions will be fixed later.
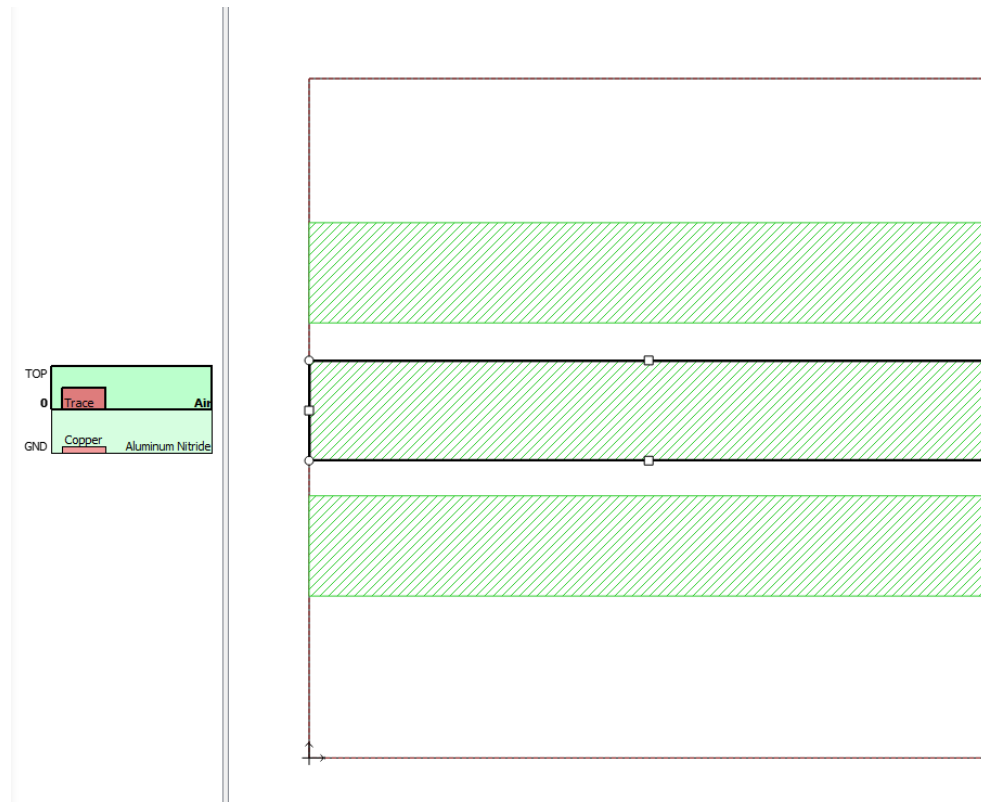


2.  Add edge ports and feedline calibration.

a.  In the top toolbar, go to `Insert/Port` and place the port on the edge of the center rectangle. Repeat for the other side.
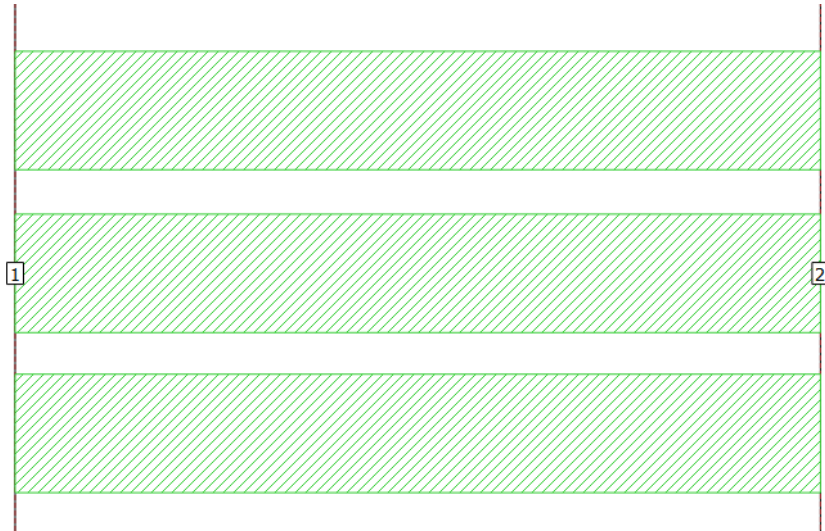


b.  Right click on one port and change the reference plane in the `Port Properties` menu as shown below. Repeat for the other side.

3. Adjust the dimensions to match those given by the calculator.

   a. Once you select a component, its dimensions will appear in the bottom right corner of the screen. You can also use the Measure tool in the right panel to measure across a component **or between them** (you will need this to adjust the separation width.)

   b. Select the center trace and drag the sides until the width is 0.054 mm. You will have to zoom in a lot.

   c. Adjust the width of the UCLs to 3.4mm. This will decrease the runtime significantly.

   d. Drag the UCL on either side of the center trace until it is 0.026mm away from the trace. Use the `Measure` tool to keep track of the distance.

   e. Select all components by dragging from outside the box and enclosing all components in the selection box. In the top toolbar, go to Object/Center and click `Vertically`. The components will center in the box.

4. Insert vias.

   a. In the right panel, select 'circular via' from the dropdown menu. Leave the number of sides at 6, and the diameter as 0.4mm.

   b. Begin entering a line of vias by copying and pasting. You can adjust the exact location of you via fence afterward, by selecting the entire row.

c. Your vias should be closer than 0.66mm from edge to edge, and around 0.8mm from the edge of the UCL.
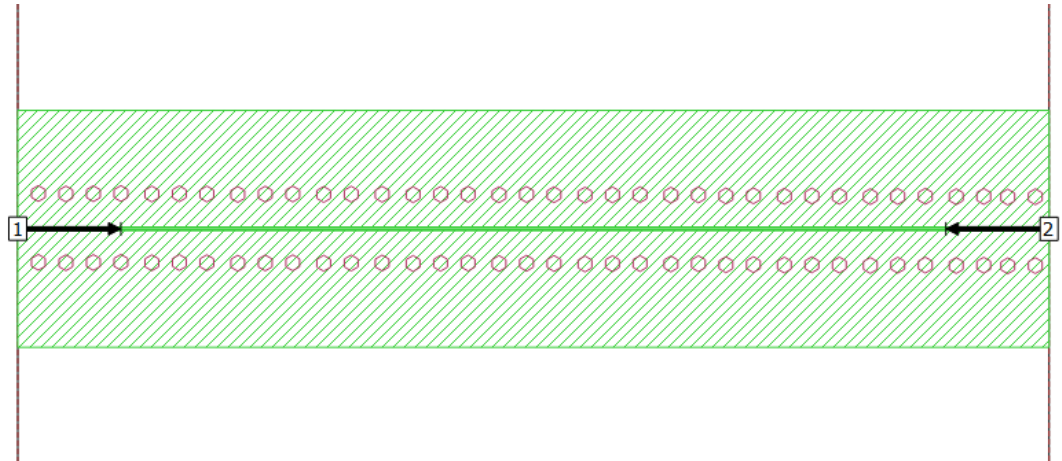


    i. My vias here are about 0.4mm apart and 0.75mm from the edges of their respective UCL.

d. Add a few more vias along the outer edges of the UCL, like this:
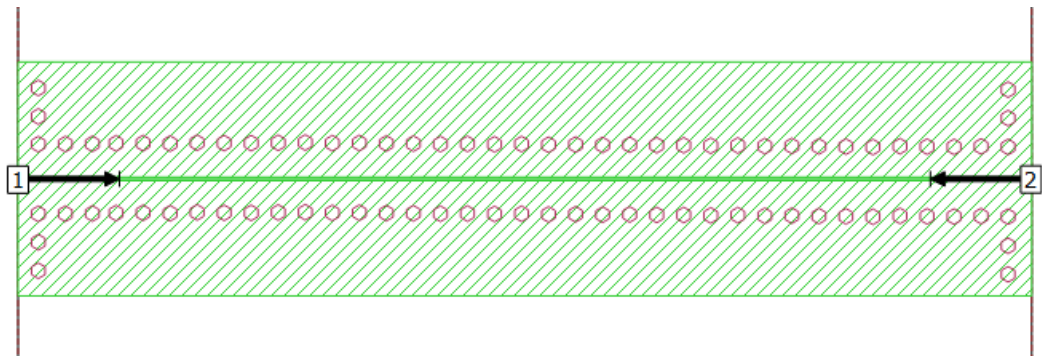


# Running Simulations in Sonnet

1. Adjust your mesh requirements.

    a. Under Settings/EM Options, ensure the slider is all the way to the left (most accurate meshing).

2. Select desired run frequencies.

a.  Under Settings/Sweeps, double click next to ABS and set the range from 1 to 20GHz (or another desired range). Click OK to save your settings.

3. Click `Add to queue` to add your simulation to the queue.

# Viewing Results/Producing Plots in Sonnet

1. Create a plot of impedance.

    a.  In the `Job Queues` tab you can monitor simulation progress. Once your simulation is finished running, click on `Graph` in the top left toolbar.

    b.  Several windows appear in the left panel: in the top `Curves` window, double click the legend entry 'MAG[PZ1]`.

    c.  To view impedance versus frequency, select `Line Z0` under `Transmission Lines`. Click Close. (You may specify the real part, imaginary part, or magnitude part in this box also).

    d.  You may change axis settings by double clicking on an axis.

2. Create a plot of reflection/transmission coefficients.

    a.  For reflection coefficient, follow the same steps for creating an impedance plot, but select `S-Parameters.` Don't close the dialogue box.

    b.  For transmission, click `Add` and change the axis format to `MAG` from `DB` and change the `From Port` setting to refer to Port 2.
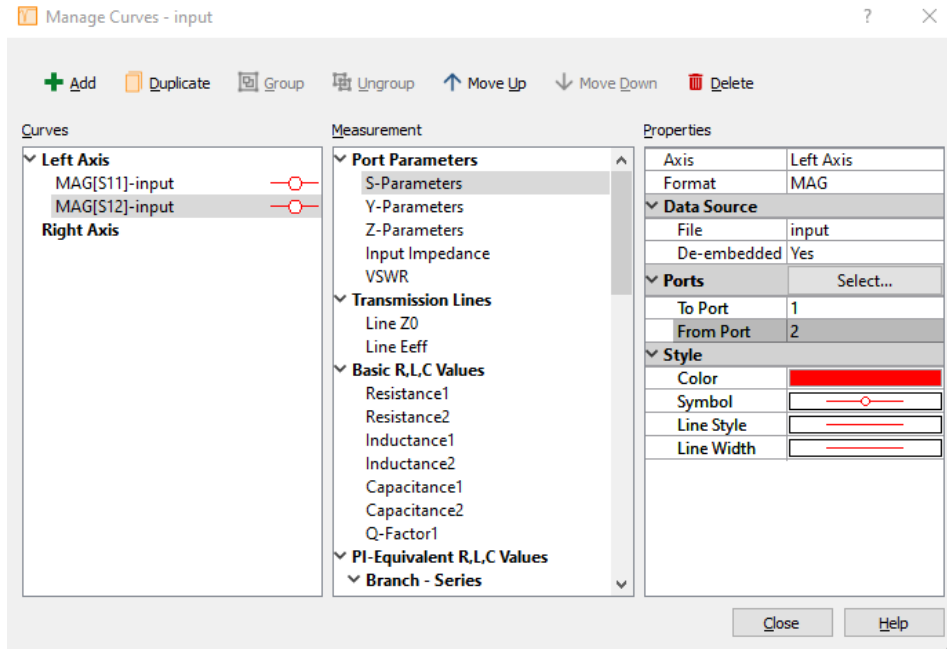
3. View current behavior in your CPW.

    a. In the `Job Queues` tab, click on your simulation in the queue and open `Currents` in the top toolbar to view currents.