Getting Started with AWR Design Environment

AWR Design Environment is a suite of simulation tools for modeling high-frequency circuits and electromagnetic structures/devices. You can create a model of a real-world physical structure, simulate the structure to check for things like matching and return loss, and then combine the model with a circuit description to simulate how the system will operate. AWR was acquired by National Instruments in 2011. The previous version of this software was called “Microwave office.” No, seriously, it really was. Thankfully they’ve since changed it to “AWRDE” (sounds like ‘award’) although you still this software suite referred to as “μWO” (micro-wave office).

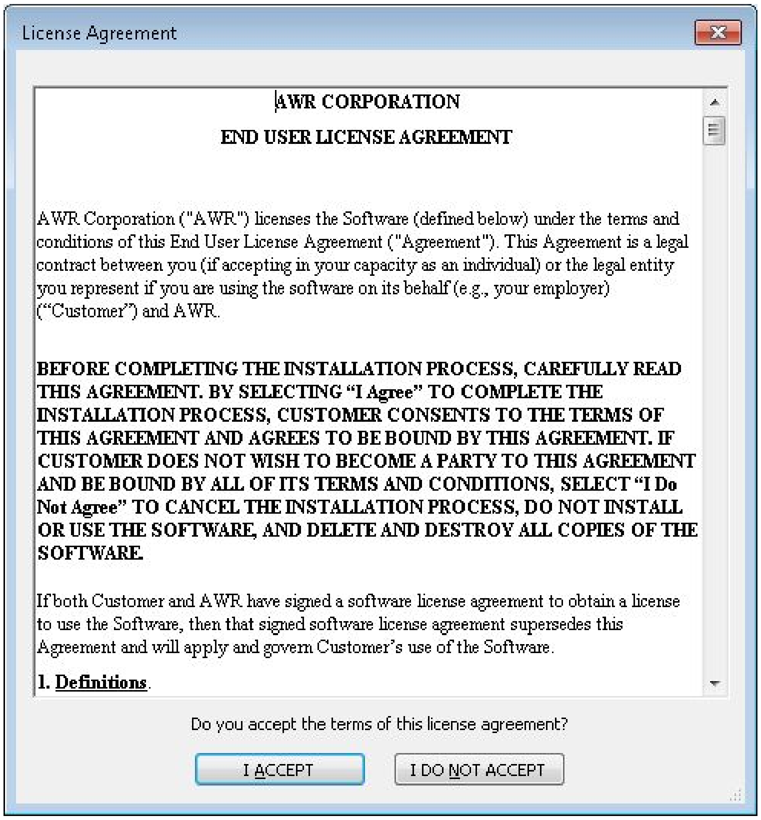
We will be using this to simulate our copper-tape antenna, perhaps use it to design a better antenna for the future, and to simulate our circuit that will convert a 915 MHz signal (the RF signal) to a DC signal that will power things like a microcontroller and other digital sensors.

In this tutorial, I will go over the basics of electromagnetic simulations using the AXIEM engine to simulate a planar antenna.

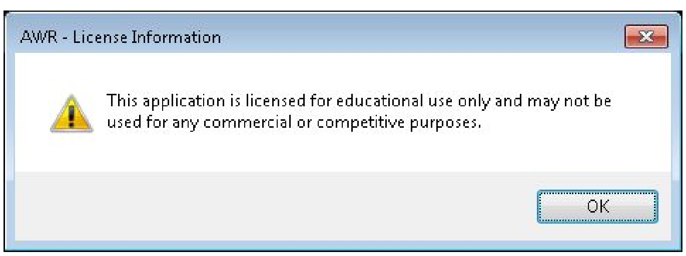
Let’s get started!

1) Double-click the AWR Icon to open the program

2) A License agreement window may open, if it does, just click “I Accept”

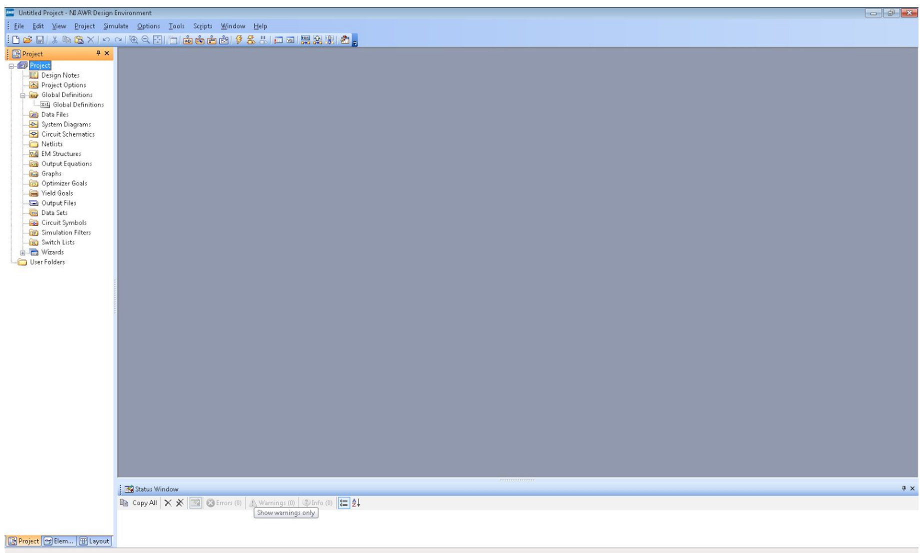


3) Click “OK” to acknowledge that we have the educational license. This windows means: if you’re gonna use this for something that someone will pay you for, then buy the real version.

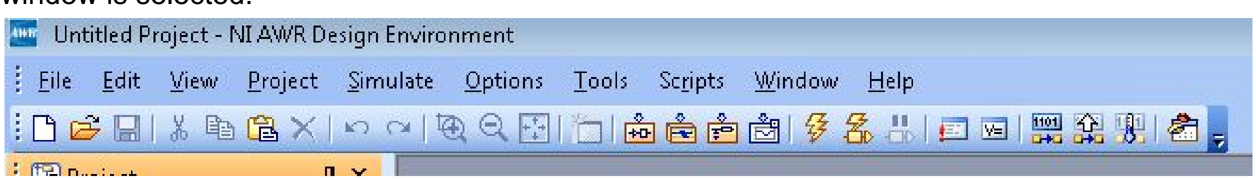


4) After a lovely splash screen,

You will be greeted with the main window of AWRDE

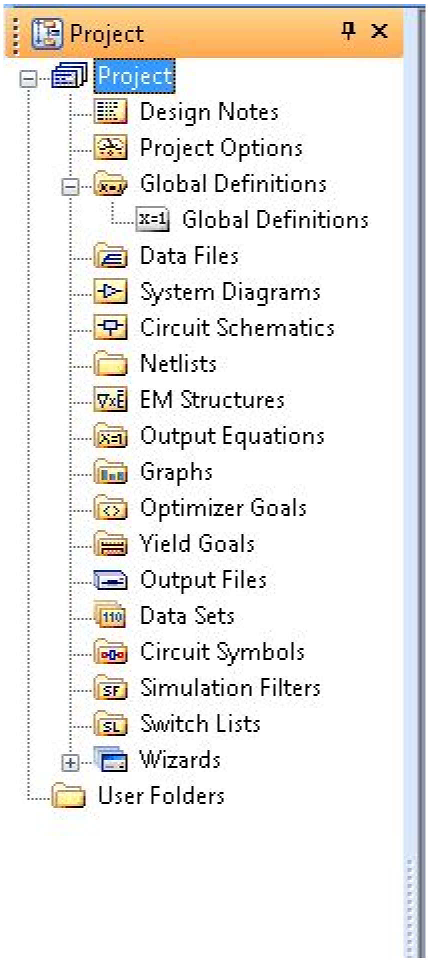


5) AWR looks much like an integrated development environment you would use to develop software with a programming language. There is the traditional menu bar (file / Edit, View, etc) and toolbar The toolbar will end up having different tools depending on what window is selected.

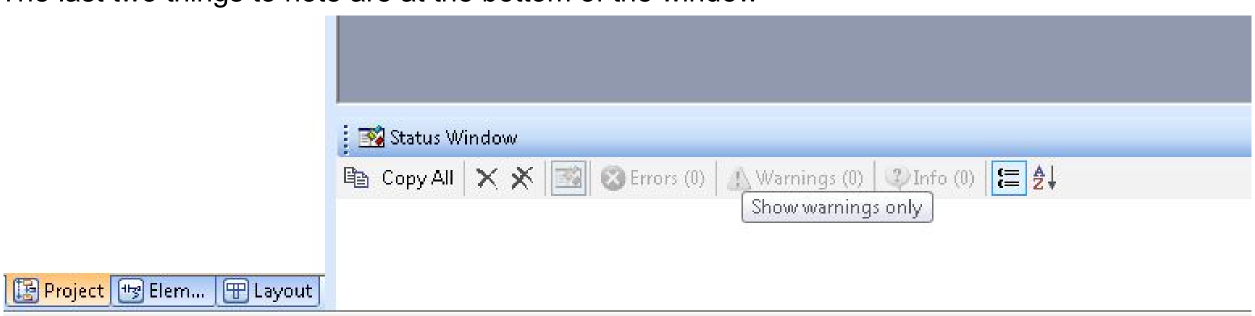


6) To the left is a large pane that has a list of categories. This is called the Project Browser. It will hold all of the different design elements that we need. You can see things listed such as “Circuit Schematics”, “EM Structures” and “Graphs.” These are the important ones. We will make either a circuit or an EM (electromagnetic) structure. We specify what data we want to be saved and how that should be displayed. The result will be put

in a “Graph” which we can then use to display our data. The other important item to see is the “Project Options” which contains global settings for our entire project we are working with. Likewise, Global Definitions can store useful variables much like a #define statement in C.



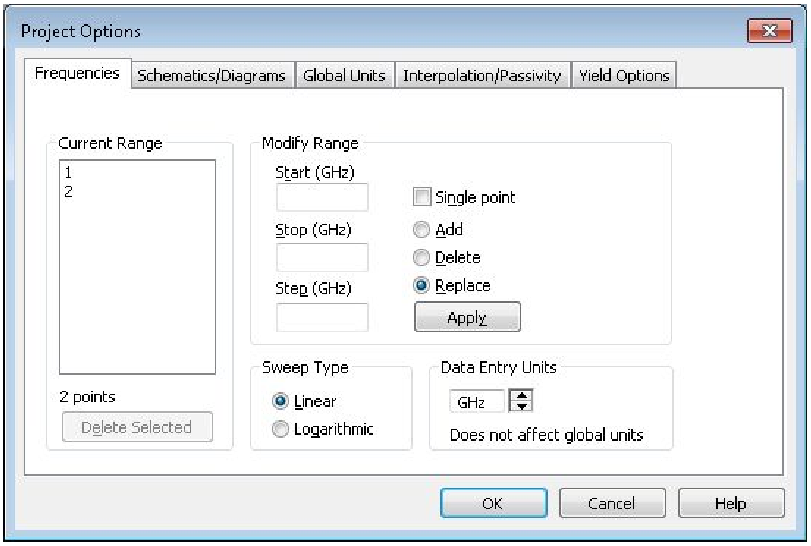
7) The last two things to note are at the bottom of the window



At the bottom of the Project Pane are three tabs that we can click. These let us cycle through various options for the current view. Since we don’t have anything open yet, they are not that useful. Later when we are building circuits and need things like resistors, power supplies, diodes, etc, these will be found using the tabs.

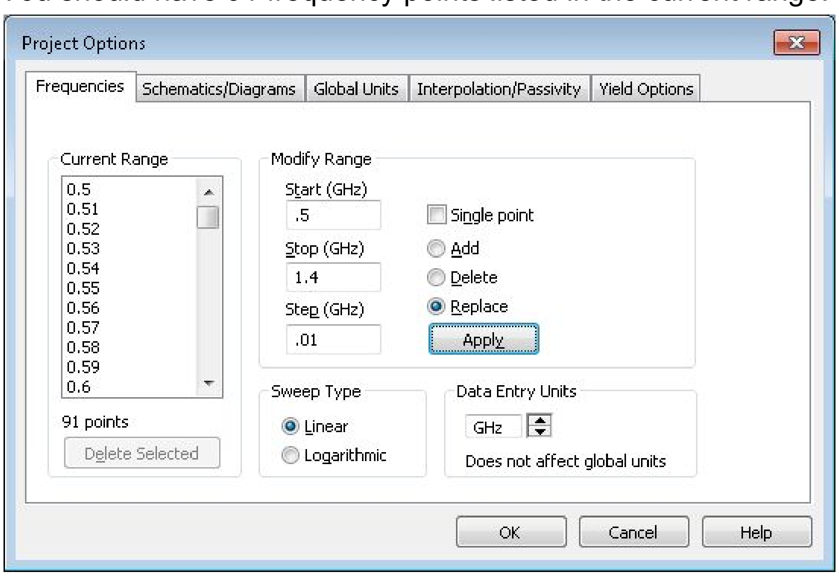
Just to the right of the three tabs is the status window. This will list various output information as you are working on your project such as warnings or errors.

8) Double-click on Project Options to see the settings.



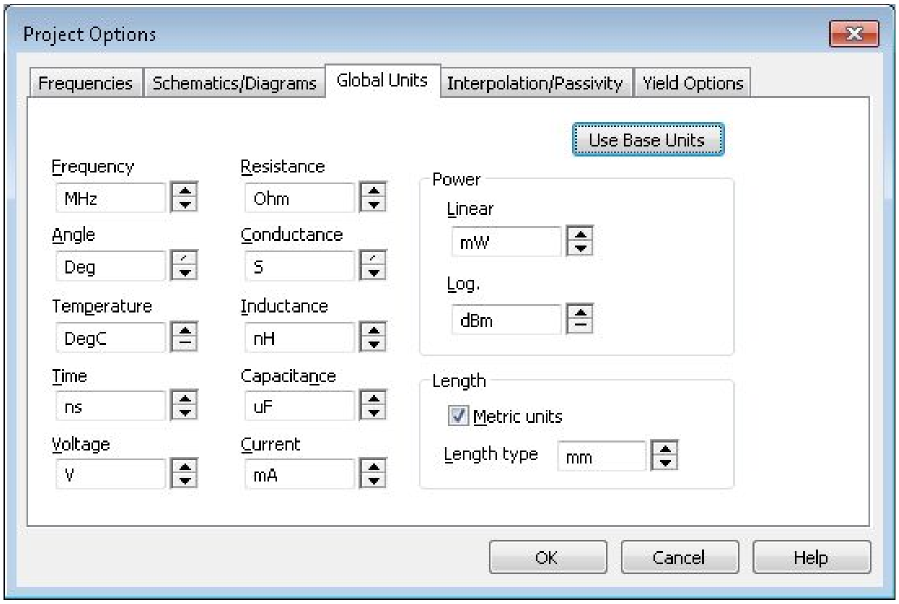
The frequency is set to be simulated at 1 and 2 GHz. Most unacceptable. Let's change this to something that might have some meaningful information for us. Since our cardboard antennas were designed around 915 MHz, let's sweep freuqency from 500 MHz to 1.4 GHz.

Set Start to be 0.5 GHz, stop to be 1.4 GHz, and Step to be 0.01 GHz. You do not need to touch the other settings, just make sure that "Replace" is selected and click Apply. You should have 91 frequency points listed in the current range.



To make our life easier, we could have changed the Data Entry Units to be ready in MHz instead of GHz. This would have been selected in the lower right corner.

9) Select the tab at the top that says “Global Units.”

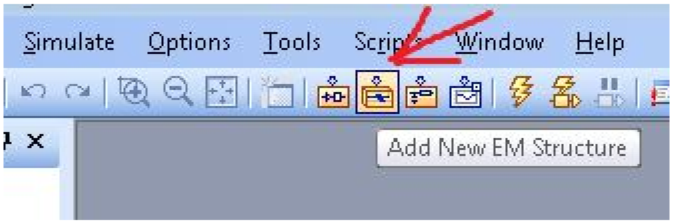


Here we see the default units of our project. Power will be in mW or dBm. Angle is in degrees, capacitance is in uF and inductance is in nH. Perhaps what is most relevant for us is to see that length type is set to mm. It would be a good idea for us to keep working in metric units. However, if we wanted to go from nice decimal-based metric land to strange, voodoo measurement US/Imperial unit land, we would unclick the “Metric Units” checkbox. When you do this, you will see the default unit set to something even weird than the inch: the mil.

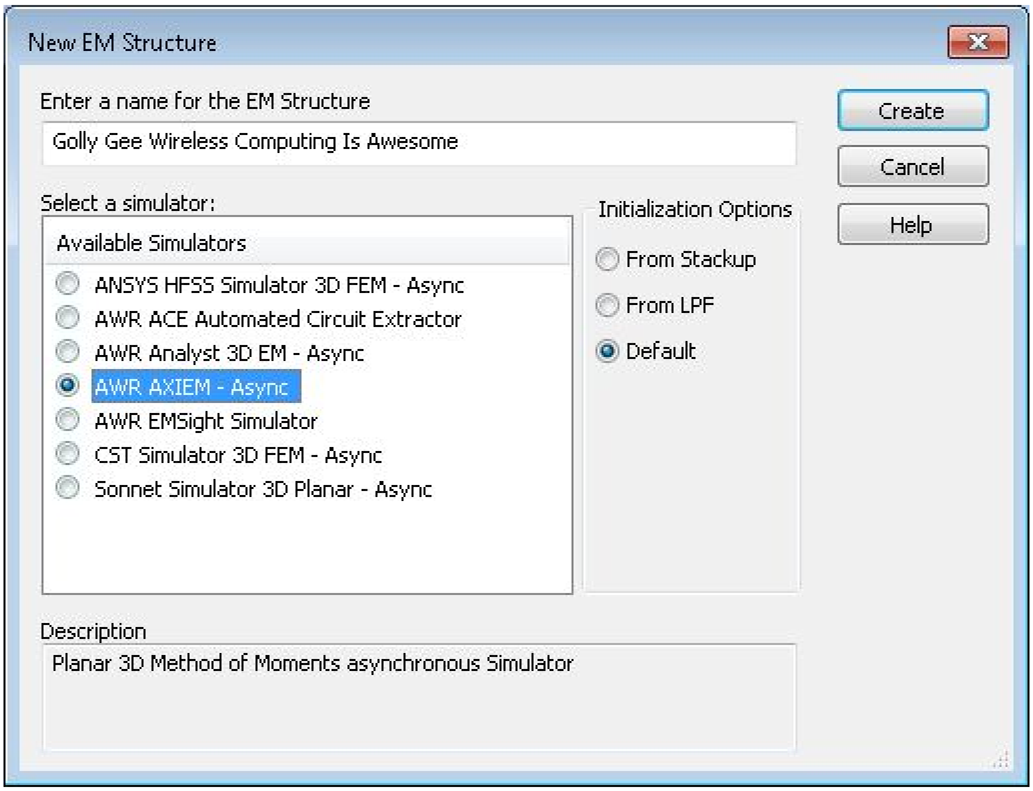
This is a thousandth of an inch. A thousandth of an already weird unit to begin with. Shake your raised fist at the sordid history of unit conversion, change back to metric, sigh, and we will continue. To remember, though: length units will be in millimeter. Not centimeters. Not meters.

There are a few other options in this settings window, but you can close it and we will go on.

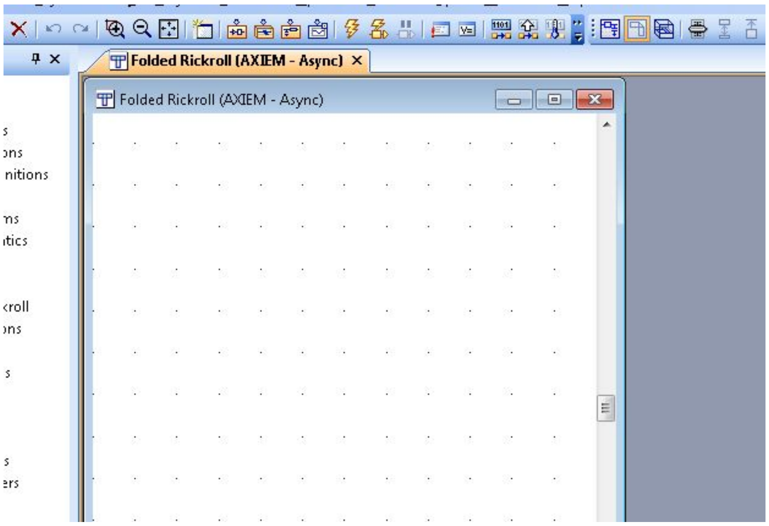
10) We are going to go ahead and make our antenna to simulate. In AWR land, this is an EM structure. In the toolbar, click the “Add New EM Structure” button to bring up the New EM structure window.



11) In this window, give your antenna a name such as “Gee Golly Wireless Computing Is Awesome” or the “Folded Rickroll.” Most important, though: **Select the AWR AXIEM simulation engine.** EMSight is old and has somewhat bad results, and we do not have HFSS or Analyst installed. Your window should match the screenshot below.

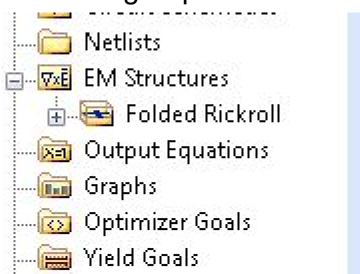


Once it does, click “Create” and the Folded Rickroll (AXIEM - Async) window will appear.



12) There are two things to note now. We have a new entry in our project pane, and some new tools on our toolbar.

In the project pane our Folded Rickroll antenna is now listed underneath the EM Structures group.

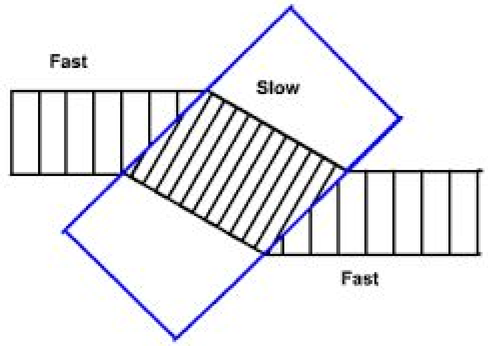


The toolbar has lots of new things such as “View 3D EM Layout”, “Substrate Properties” and a few others which I’m sure you can move your mouse over to and look at just as easily as you can read this.

The View 3D EM Layout button is pretty useful, but for now there is nothing to show. We first need to describe our structure.

13) I don’t want to get into a long discussion, but in short, the AXIEM simulation tool will simulate the electromagnetic properties of the geometric structure that we describe. This is going to be CAD type drawing where we define rectangles, polygons, etc.

What’s most important, though, is knowing what kind of conductive material is on top of what type of non-conductive material. Back in an earlier lecture, we saw that a unit step function ended up looking like a star-step function because of energy traveling down a line and being reflected back. We measured the time it took for that wave to travel down and back again through the cable. This is a measurement in meters per second and we found the speed of light. However, this speed was not the 3 x 10^8 m/s that we normally see. That's because the 3 x 10^8 constant is the speed of light in **vacuum**. The speed of light will actually change depending on the surface where it is traveling. This is the reason we can shine a light in water and it looks like it completely changes direction.



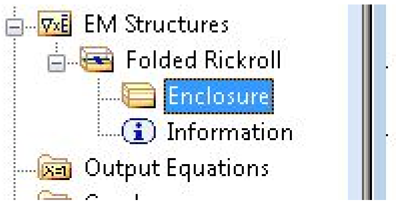
Because the speed of light in these materials is different, the light refraction occurs and things just look all bent up and stuff (to be technical, of course).

Well, an EM signal is just another form of light, the speed which the wave propagates is going to vary depending on the material. Typically, a copper trace (which has a very high conductivity) will be used to carry an EM signal. The copper trace is placed on a PCB substrate. These cay vary, but are often made from a material called FR4. This material has a certain speed of light associated with it and the signal traveling down the copper trace will be bound by the speed of light in the FR4 material that the copper is printed on.

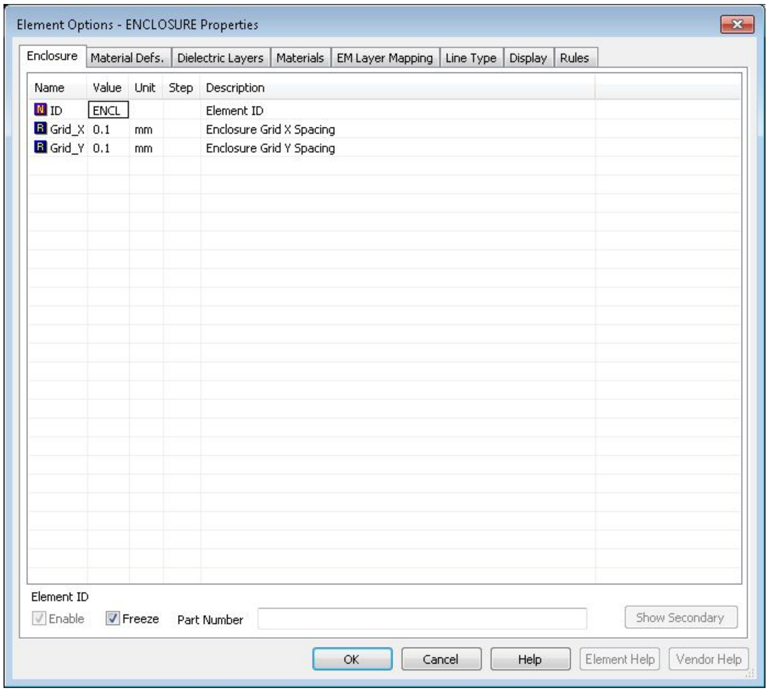
That's my spiel. You need to know the "Dielectric permittivity" of the material so that you can simulate it. You've measured antennas. You would not expect an antenna to work the exact same way as when i am holding it in the palm of my hand as you would if it were dangling in thin air. The AXIEM engine needs to know these types of things so it can simulate the antenna.

The (relative) dielectric permittivity for corrugated fiberboard (cardboard) is about 1.78 with a loss tangent of .018. (I found these values someone else measured from the Internet). Our cardboard is about 3.8 mm deep. We used copper tape that was placed on top of the corrugated fiberboard. You will need these values for the next step.

14) Click on the itty-bitty plus sign next to your folded Rickroll antenna to expand its contents and double-click the Enclosure entry

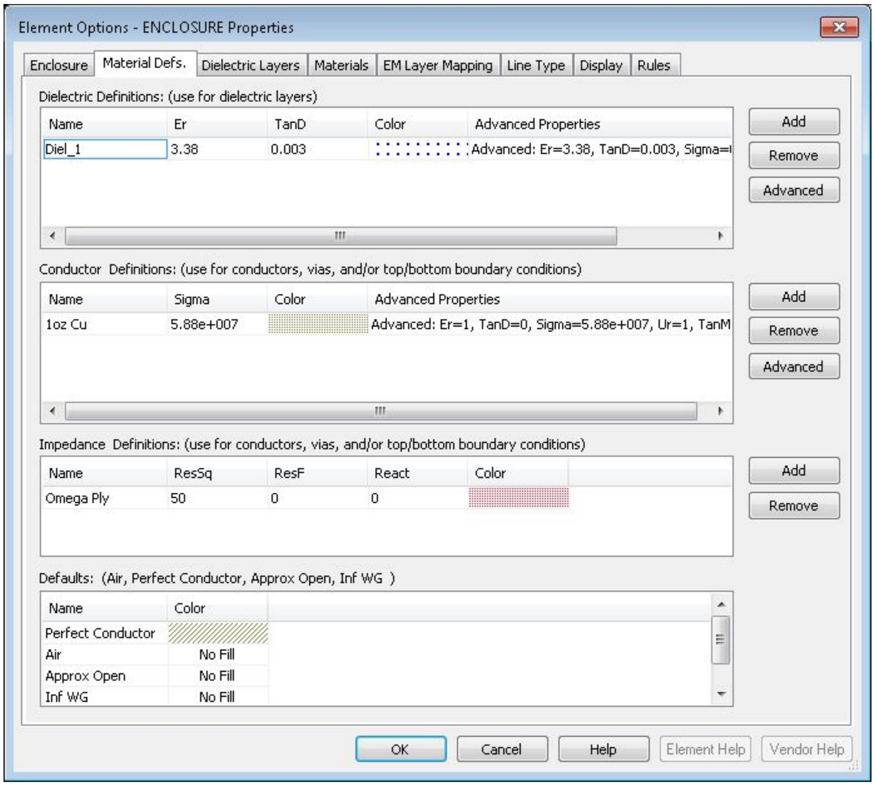


15) This will bring up the Enclosure property window.



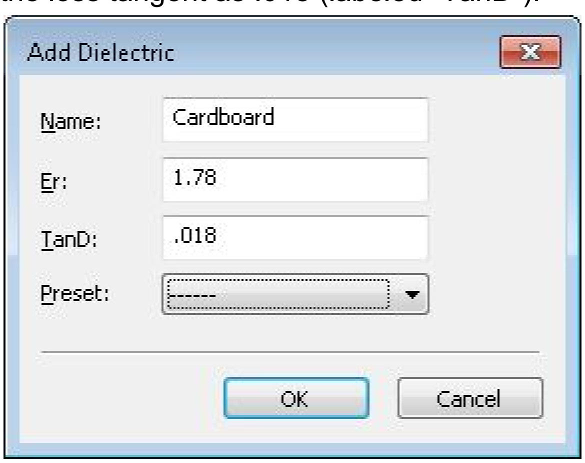
16) The Grid_X and Grid_Y settings specify the grid where we can place our objects. All verticies of the 2D polygons and shapes that we specify must fall on a point in our grid. We will not need to change these, but keep these setting in mind.

17) To set all the lovely settings we need, click over to the Material Defs. tab.

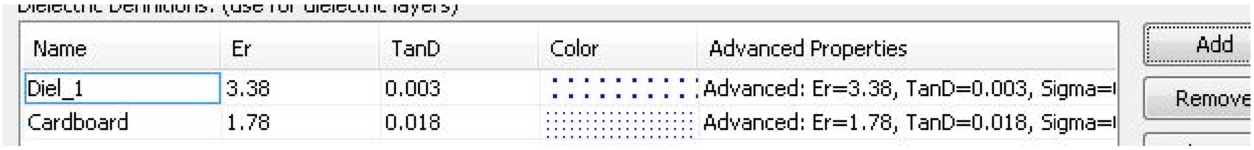


This defines all of the different materials that we have. We are going to be using a cardboard delectric layer (the top box) with a copper conductor (the 2nd box) and we don't need to change anything about the last 2 boxes.

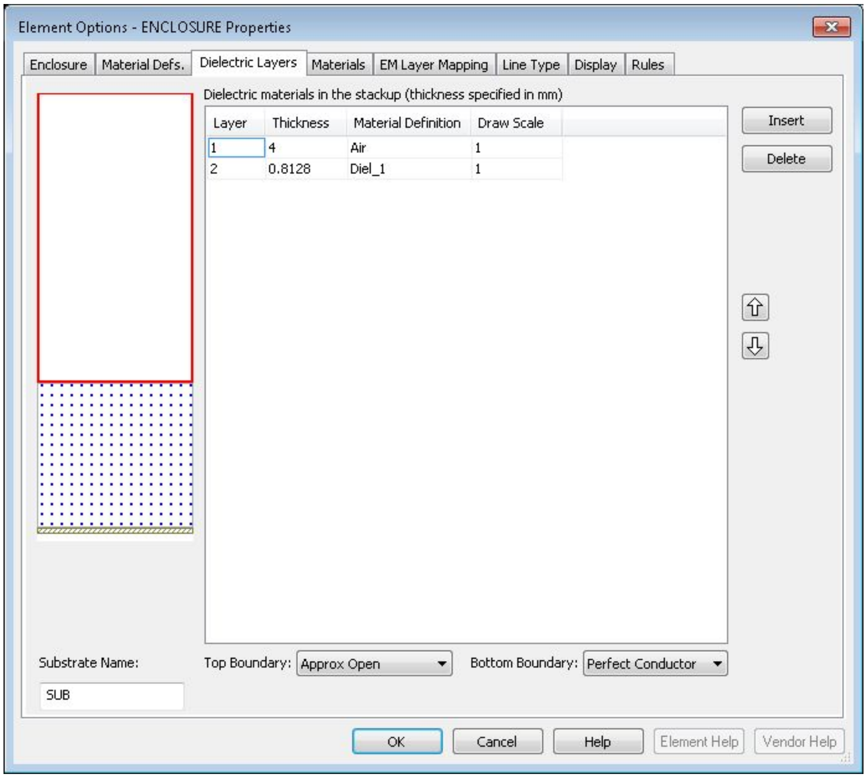
18) To add the cardboard substrate, click Add to open the "Add Dielectric" window. Enter the name as Cardboard, specify the releative dielectric parameter as 1.8 (labeled "Er") and the loss tangent as .018 (labeled "TanD").



Hit OK, and the new dielectric layer will appear in the top Dielectric Definitions box.



19) Now, select the Dielectric Layers tab.

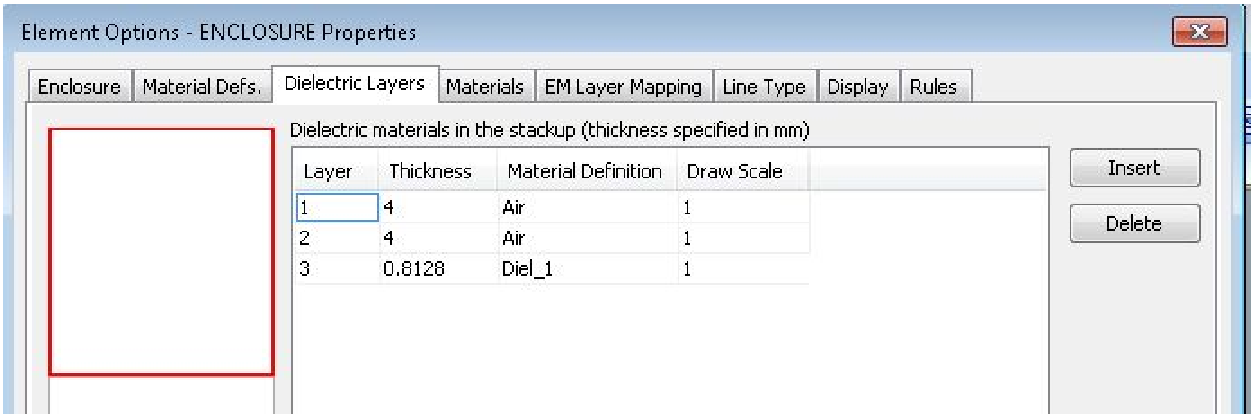


Here, we map how the dielectric layers we have are mapped to reality. The simulation defaults to having a perfect electric conductor on the bottom layer, with a substrate on top of that, and then air extending above. This kind of makes sense for most applications. Imagine a circuit board. On one side is typically a layer of copper connected to ground. This is called the "ground plane." The opposite side of the board has our copper traces that we would put on.

But this is all bananas for us. We want to have nothing below us (ok, technically the earth is below is, but we can ignore that for this experiment) but air, have a plane of cardboard, have some copper on that and then more air above us.

20) To do this, we need to specify the top boundary of our simulation to be "Approx. Open." Good thing that's already set for us! We need to make the Bottom Boundary match. So do just that.

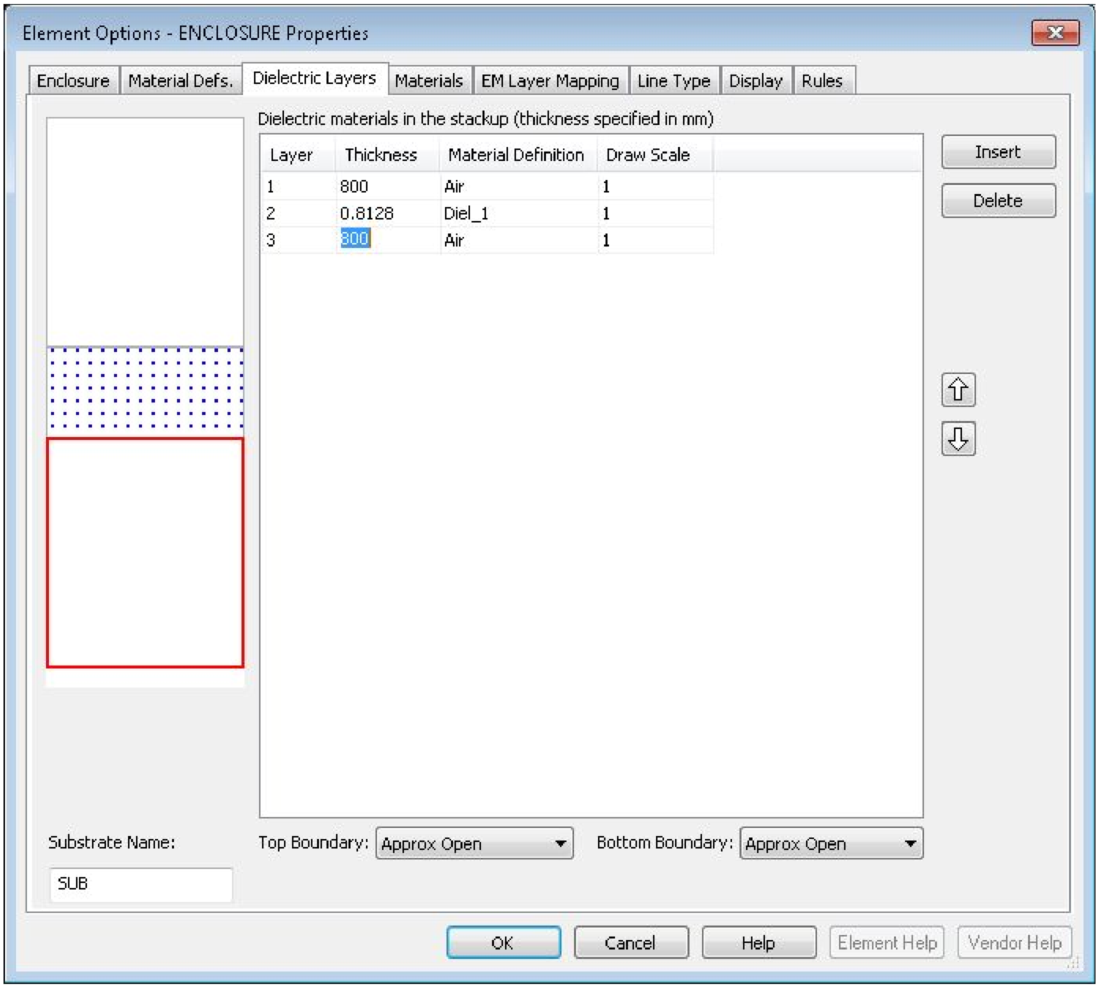
21) Next we need to give an air gap below our cardboard. With Layer 1 (the top air layer) selected, click the "Insert" button. This will copy the air layer to a new layer.



Click the down arrow on the right to lower one of the air layers to be at the bottom. It should eventually look like a dielectric and air sandwich. (Air over dielectric over air).

22) But how thick is our air? Ok, not really thick is in musty or smelly, but how far should our simulated air layer extend above and below our cardboard. This isn't too important, but just give it a big number. I said 800 units above and below. What are those units? mm.

Right now, your window should appear like mine:

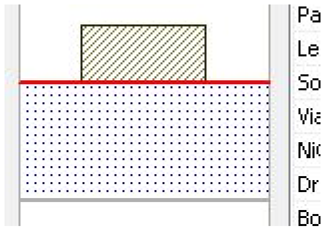


23) Now we need to specify that our middle dielectric layer is made of cardboard and is 3.8 mm thick. For layer 2, change the material definition to be cardboard and the thickness to be 3.8. Almost done with setting up our materials!

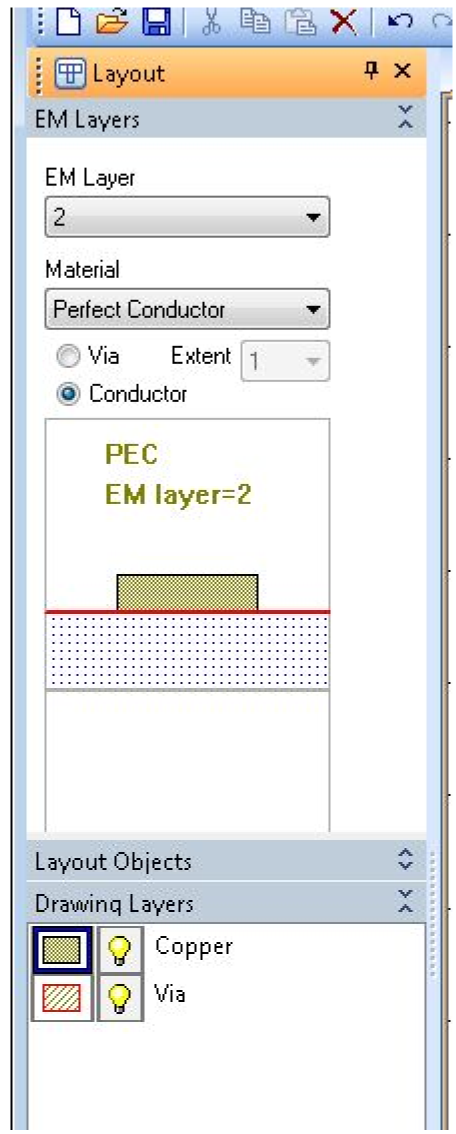
Dielectric materials in the stackup (thickness specified in mm)

Layer	Thickness	Material Definition	Draw Scale
1	800	Air	1
2	3.8	Cardboard	1
3	800	Air	1

24) Click the EM Layer Mapping tab. Click on the element labeled “Copper.” You should see a picture showing a big square of copper on top of our cardboard layer shown in the left of the window. Yep, that sounds exactly right. In this window, we set where physically conductive layers will be placed. There can be multiple layers of metal, but all we care about is making sure that the Copper layer is on EM Layer2. Once you’ve clicked a few things and just looked at what all is here, go ahead and hit OK to close the ENCLOSURE properties window.

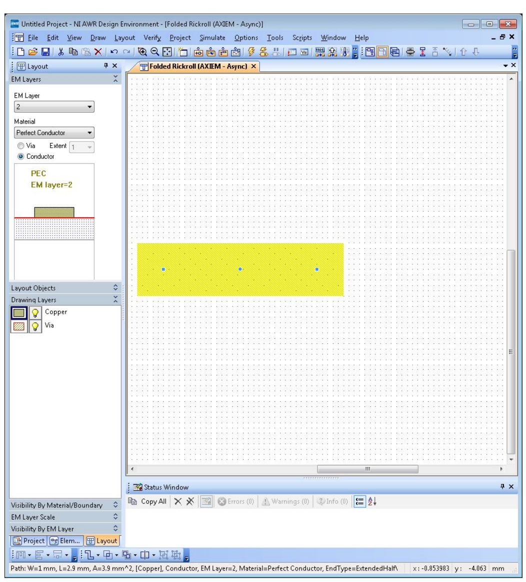


25) We have the enclosure set, so now we begin placing some tape. Click the Layout tab from the bottom of the Project Pane. We want to draw on EM Layer 2, so select this layer, use “Perfect Conductor” select the Conductor option instead of the VIA option. With this selected, you can draw a shape.

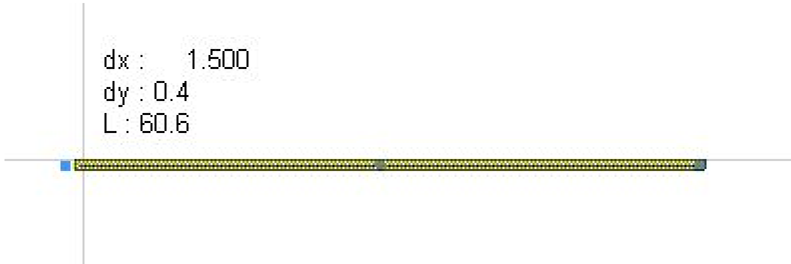


26) The shape buttons are in the lower left of the AWR window. Select the Path tool. To draw a path, first click where we want to start, then click endpoints of the path until you are satisified. Then hit Enter to draw your path.

For us, click once in the middle of the screen, and then move your mouse left (you may need to zoom out using Control + mouse wheel) to and click to make a horizontal line. Hit enter, and you should see your brand new copper tape (modeled) path.

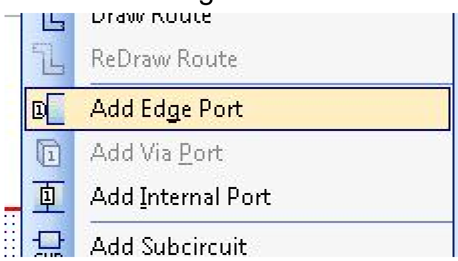


27) Zoom out a lot and drag the left node of the path until the length is about 60 (L=60, or within 5 mm should be seen)

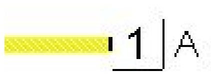


28) Now we need to define where our feed point is. In other words, where is our 50-ohm cable going to be connected to the copper tape. In AWR, feed lines can only be connected to the edge of a polygon. So if you want to put a feed point in the middle somewhere, you'd need to draw a new shape inside of the larger shape and place the feed there. We are going to place our positive terminal to the right of the copper strip and our ground terminal to the left.

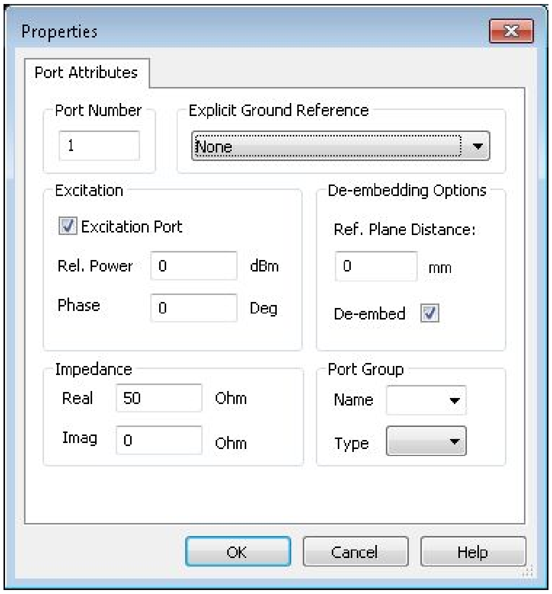
29) Click the copper shape to make sure its highlighted and from the AWR menubar select Draw > Add Edge Port



With the new pointer, click the right edge of the line. This will place a port on that edge.

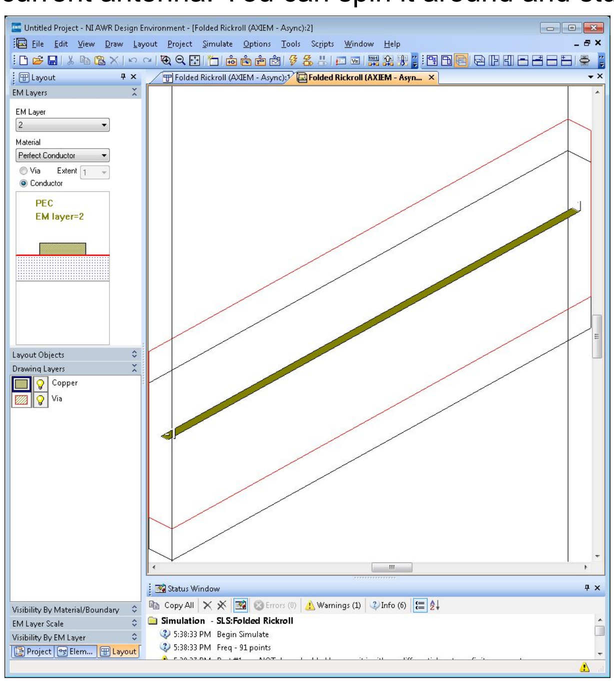


30) Double click the port to bring up the properties menu. This menu contains the options for the port. Note that this port models a 50-ohm cable, just like our real system we are using. The number of the port is in the upper left of the window. This is port 1 for our system so far. Change the Explicit Ground Reference for this port to be "None." Hit OK to close the window.

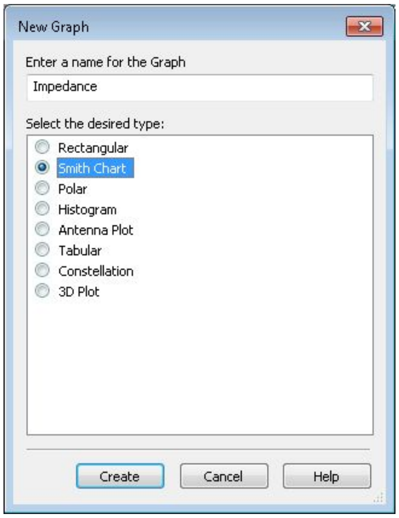


31) Repeat this process to add an edge port to the right side of the copper strip. Now, when you double-click the port to bring up the properties window, give it a port number of “-1”. This will make the port on the left edge the ground reference location for port 1. Hit OK to close the window.

32) At this point, if we click the View 3D EM layout, we will have a fancy 3D view of our current antenna. You can spin it around and stuff if you want.

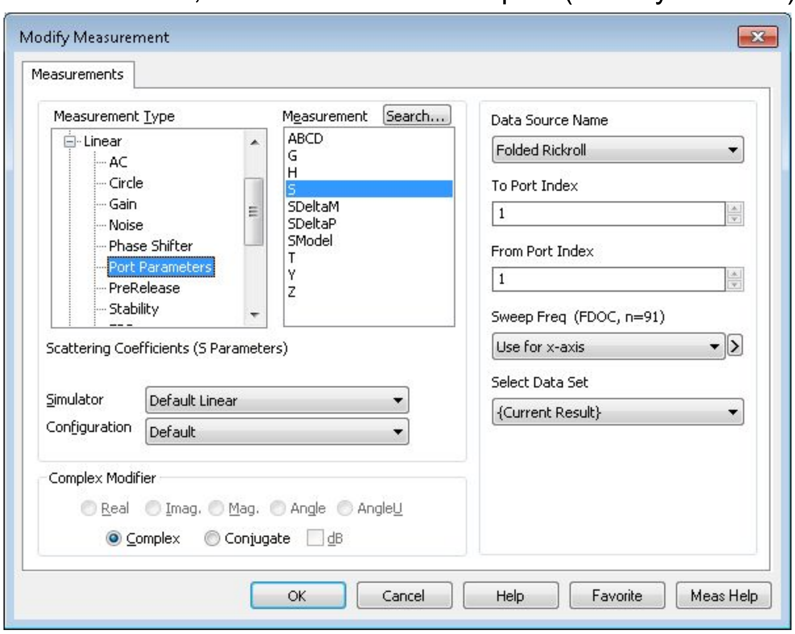


33) Go back to the main Project Pane view, right click on the “Graphs” group and select New Graph. Type a name of Impedance and select a type of Smith Chart, and select “Create.”

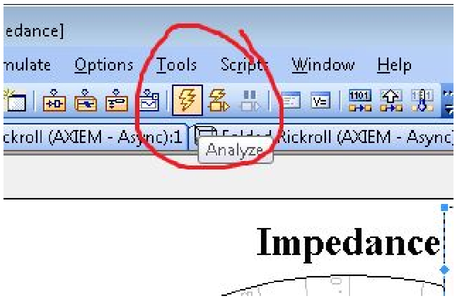


This brings up a nice looking Smith chart, but there’s nothing plotted. Right click on the plot and select Add New Measurement.

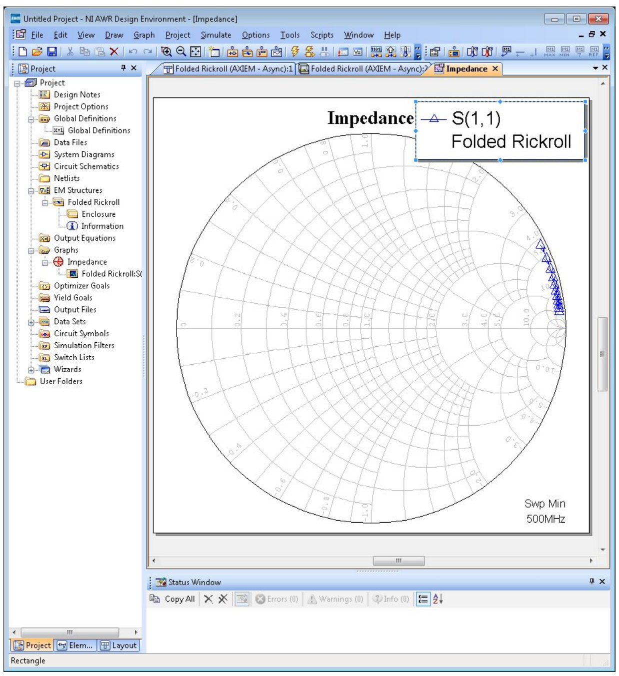
The default options should be mostly fine. We are adding a Port Parameter measurement type with the S measurement. Select the Data Source Name to be our Folded Rickroll, show the data as Complex (already selected) and select OK.



34) We still have no data, so we need to run the simulation. This is the awesome looking lightning icon. I have no idea why it is lightning, but it’s really fun to click.

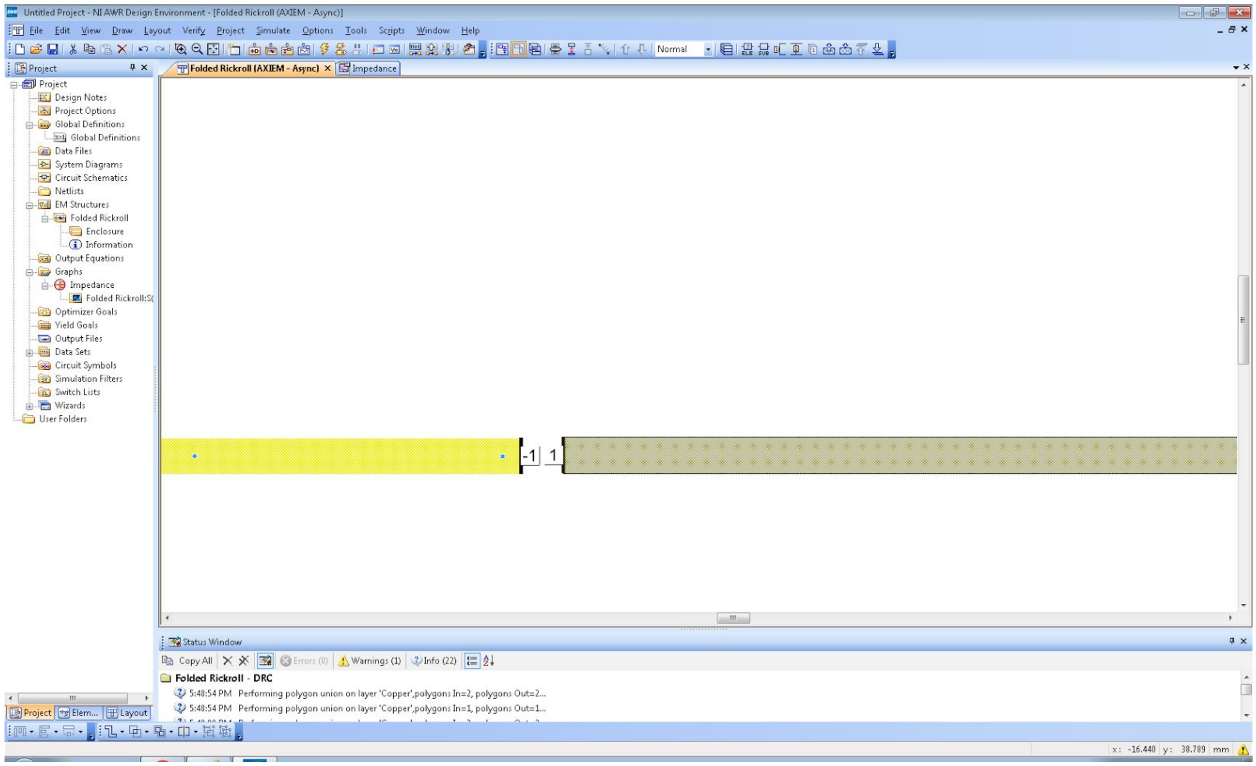


Click this button, and after some windows come up, we should see some horrible looking data.



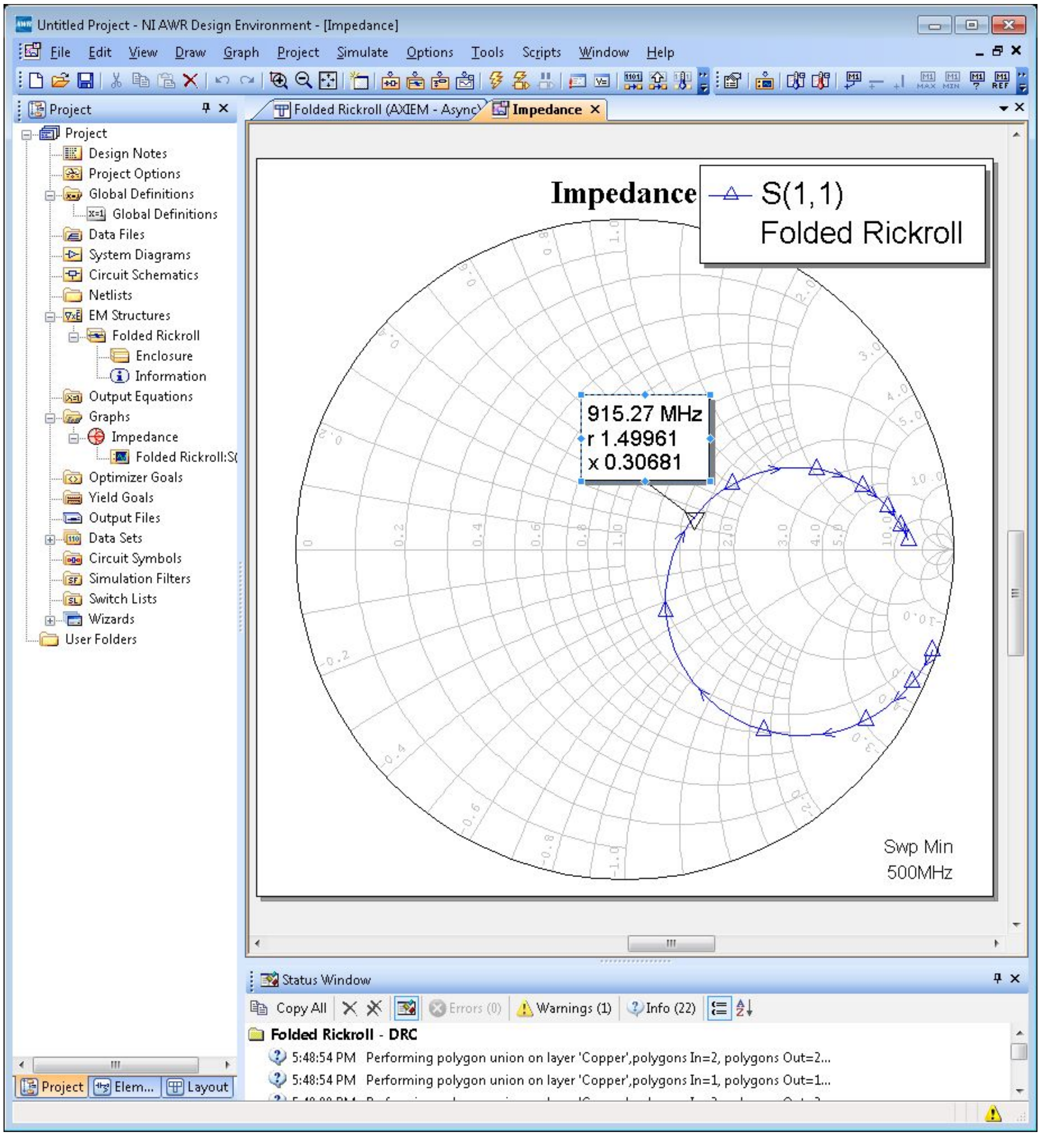
35) Woo hoo! We simulated our device! Now we just need to modify our geometry so it looks good.

36) Go back and modify your structure until you find something that crosses the real line. Hint: For myself, I made a dipole out of two lengths of copper. The feeds were in the center with the ground side mirroring the other side. My geometry is as follows:



With the lengths roughly at 60 a piece.

My Smith chart looks like this. I have added a marker to point out where 915 MHz is.



Be sure that you know how to do this. Next lecture, we will build on this and add in return loss and 3d radiation patterns.

Go ahead and start inputting your group's antenna model into the AWR system. We will simulate this and see how well it matches up with our measurements we took.