

# **PCB PARASITIC INDUCTANCE ANALYSIS USING ANSYS Q3D**

Dharini Parthasarathy  
Updated by : Nick Wolford

Electrical and Computer Engineering Department, Miami University,  
Oxford, Ohio, [parthad@miamioh.edu](mailto:parthad@miamioh.edu) wolfornd@miamioh.edu

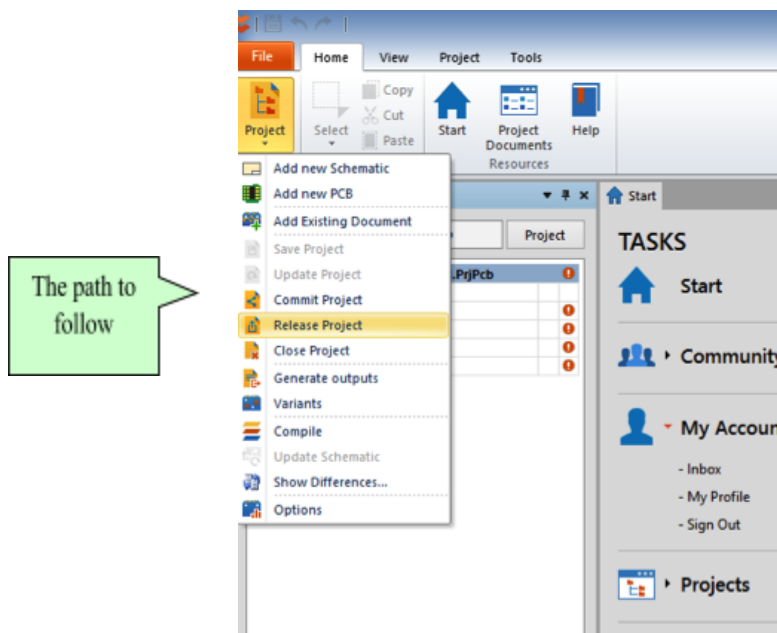
## TABLE OF CONTENTS

<b>TITLE</b>	<b>PAGE NUMBER</b>
Section 1: Release Project in CircuitMaker	3
Section 2: Altium Designer	6
Section 3: Exporting as a DXF file	9
Section 4: Importing DXF file into Q3D	11
Section 5: Creating the 3D Model	13
Section 6: Nets and Excitations	24
Section 7: Reducing Matrices	27
Section 8: Creating a Solution Setup	29
Section 9: Analysis Considerations and Meshing Operations	32
Section 10: High Performance Computing Options and Setup	34
Section 11: Validation and Analysis	37
Section 12: Results	38

## Section 1: Release Project in CircuitMaker

If you have created your schematic and PCB in CircuitMaker, there are certain steps you need to take in order to import the file into Q3D. The first of these steps is to Release your Project. A Released Project is a snapshot collection of all files in the Project, and includes a choice of generated output files such as schematic/PCB printouts, part lists (BOM), fabrication and assembly files, etc. The files are bundled as a unique revision of the Project, stored as a single ZIP files, and preserved in the Community Vault as a named and date-stamped revision of the Project. To Release you Project:

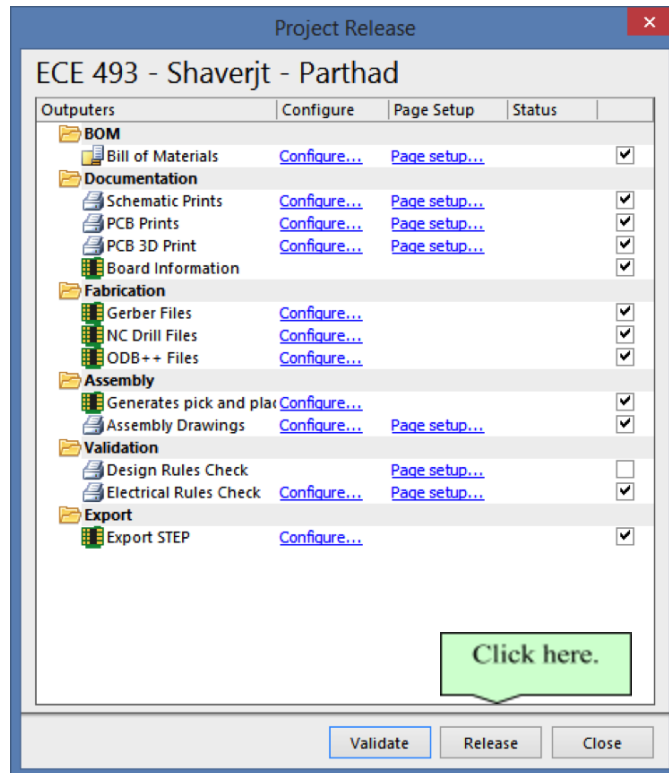
1. Open and Login to CircuitMaker.
2. Open your Project.
3. Click on *Open Design*.
4. Click on *Project > Release Project*.



**NOTE: If the project is not Saved and Committed, Release Project will appear faded.**

5. Place a check next to all the Outputters you want, and then click on *Release*.

This Project Release dialog box will appear after you click Release Project.

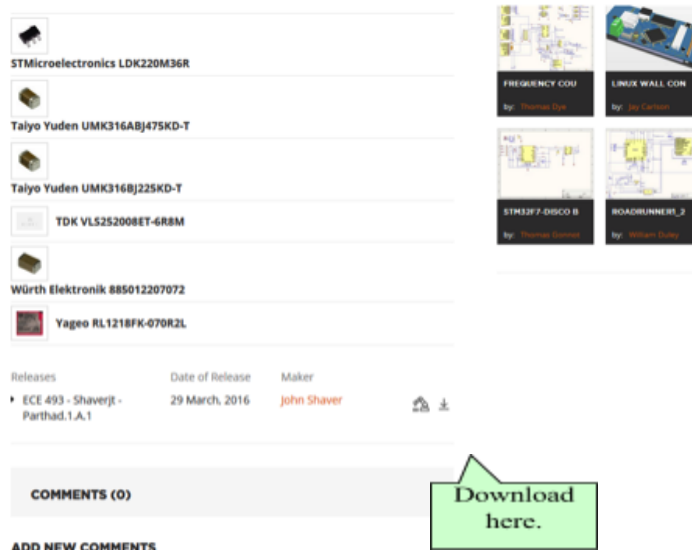


**NOTE:** Using the Validate button will authenticate the Output files

**NOTE:** After the files are Released, a Release Summary dialog box will appear. Click Close to exit the box.

6. Click on *Components & Releases* on your Project.

7. Under 'Releases,' you will see the Project you released. Download the released Project.



**NOTE:** You will have to scroll down past the components for your Project to see your Releases.

**NOTE:** All your Releases will appear here uniquely named, dated and time-stamped. Expand an individual revision to see its included design and output files.

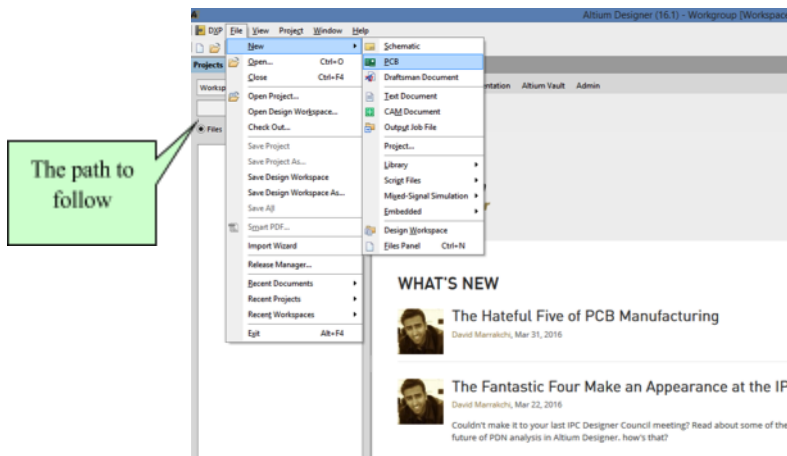
8. Rename and Save the Project in a directory of your choice.

In this downloaded Project folder, the PCBs will be saved as CMPcbDoc files. To import these into Q3D, they need to be saved as DXF files. For this conversion, you will need Altium Designer.

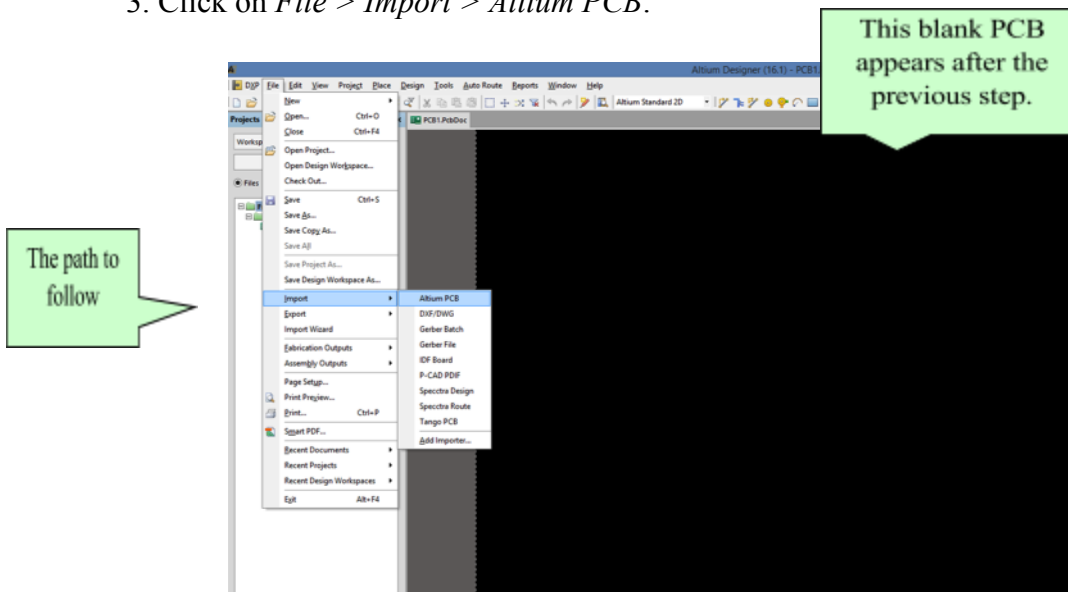
## Section 2: Altium Designer

In this section, we will open the CMPcbDoc file that was the Released output from CircuitMaker in Altium Designer.

1. Open Altium Designer.
2. Click on *File > New > PCB*.

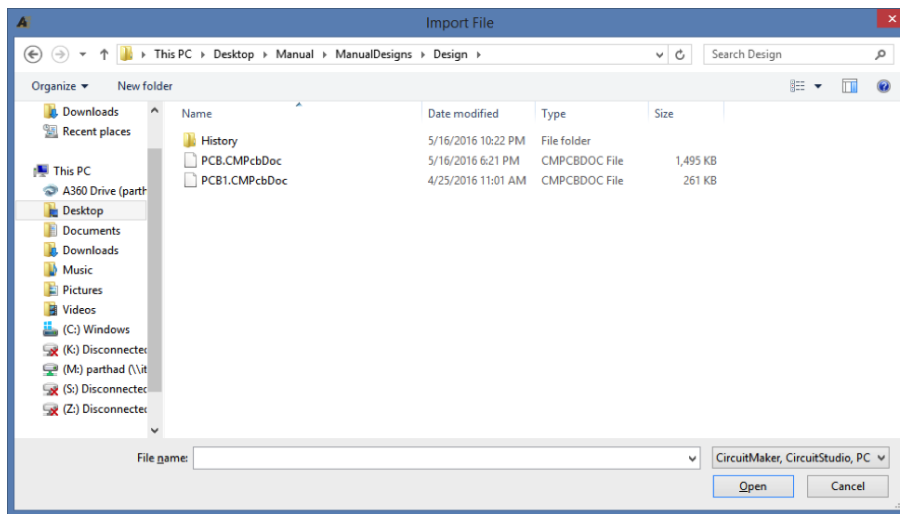


3. Click on *File > Import > Altium PCB*.



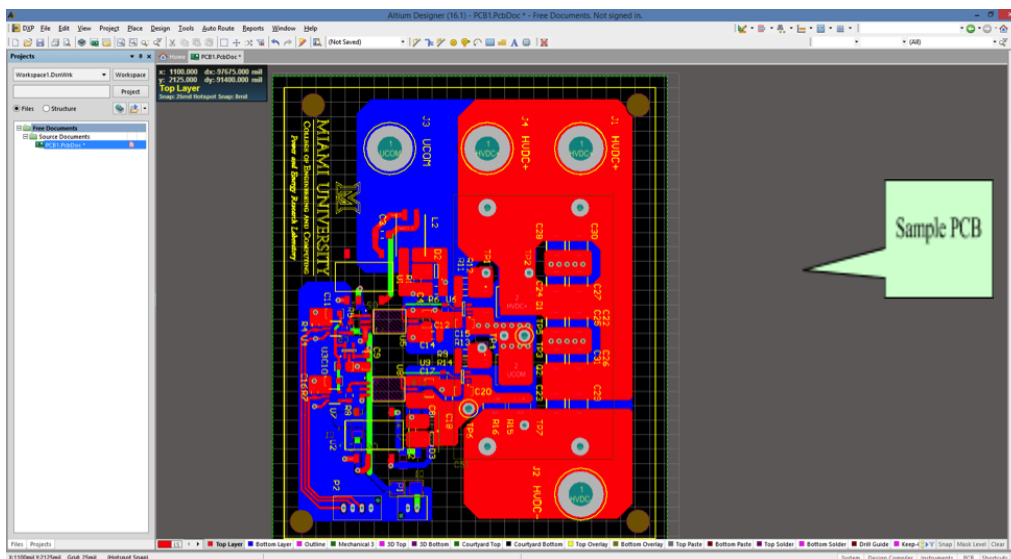
4. Navigate to the folder you downloaded from CircuitMaker, and then into the Design Folder.

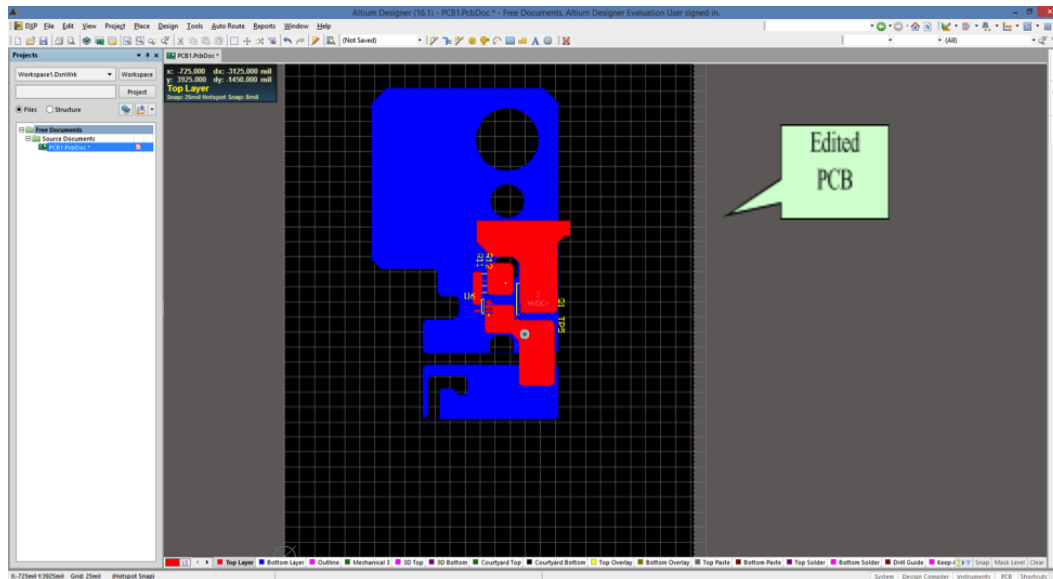
5. Open the CMPcbDoc file that contains your PCB layout.



**NOTE: Remember that the project you downloaded is a ZIP file. Unzip the file to be able to navigate to it.**

6. Edit the PCB.





Here, we are testing the gate drive loop of this PCB. So, it is edited down until only the traces that are a part of this loop remain. The more traces and pieces that you leave in, the more accurate that your simulation may be, though if there are very skinny or oddly shaped pieces it may cause the simulation to be less accurate or fail later on.

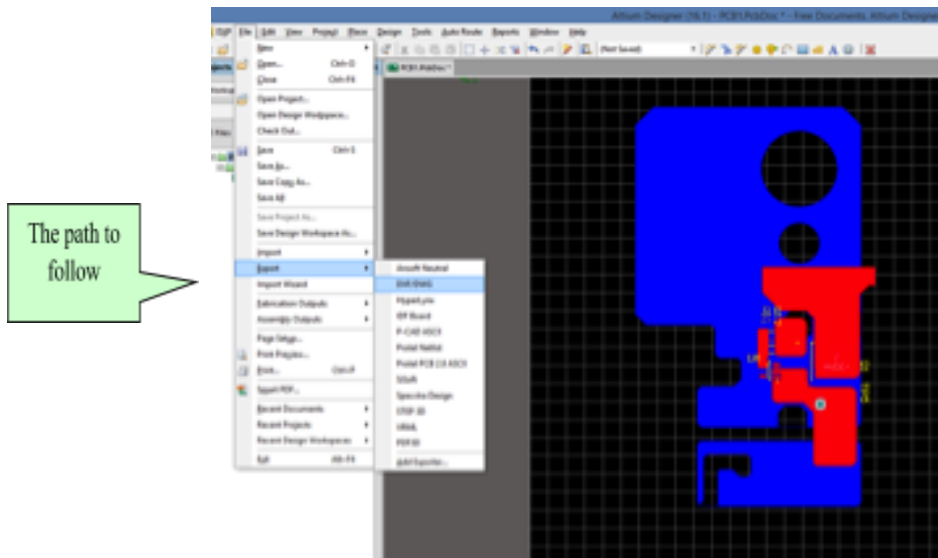
Optionally, you can wait until you import the file into Q3D to edit it down to the essential components, however this may take longer due to the greater number of objects that Q3D will create during the import process.



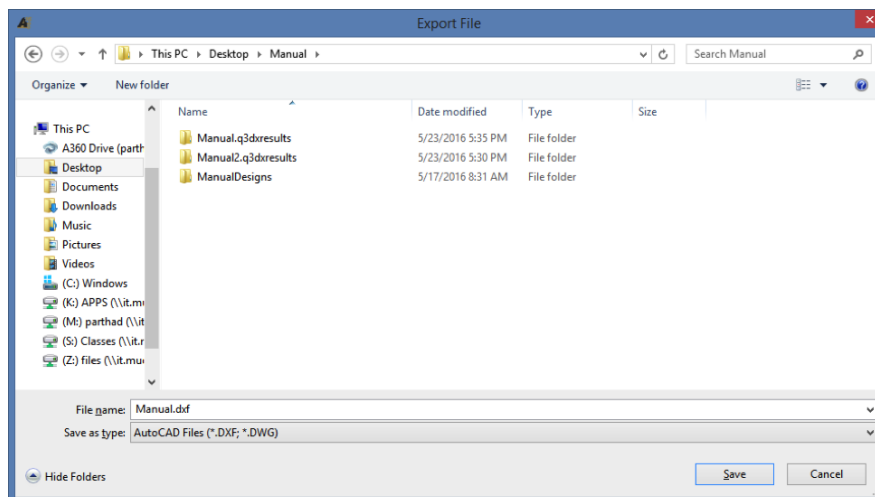
### Section 3: Exporting as a DXF file

Once the PCB has been edited to only include the traces and components that are a part of the loop under analysis, we will export the PCB as a DXF file.

1. Click on *File > Export > DXF/DWG*.



2. Rename and Save the export in a directory of your choice.



3. Select the proper format for the export and click OK.

Select 2010  
Version.

Export to AutoCAD

Setup

Options

AutoCAD Version: 2010 Format: DXF Units: Imperial

Holes

☒ Export pads and via holes On dedicated hole layers

Pads and vias holes will be exported. A pad hole layer and via hole layer will be created in the exported file. Pad and via holes will be exported on that layer.

Components

☒ Export as blocks  
☐ Export as primitives

Tracks and Arcs

☐ Export with rounded ends  
☒ Export with square ends

Primitives To Export With Zero Line Widths

☒ None  
☐ All  
☐ Selected

Layers

☒ All  
☐ Currently Visible Layers

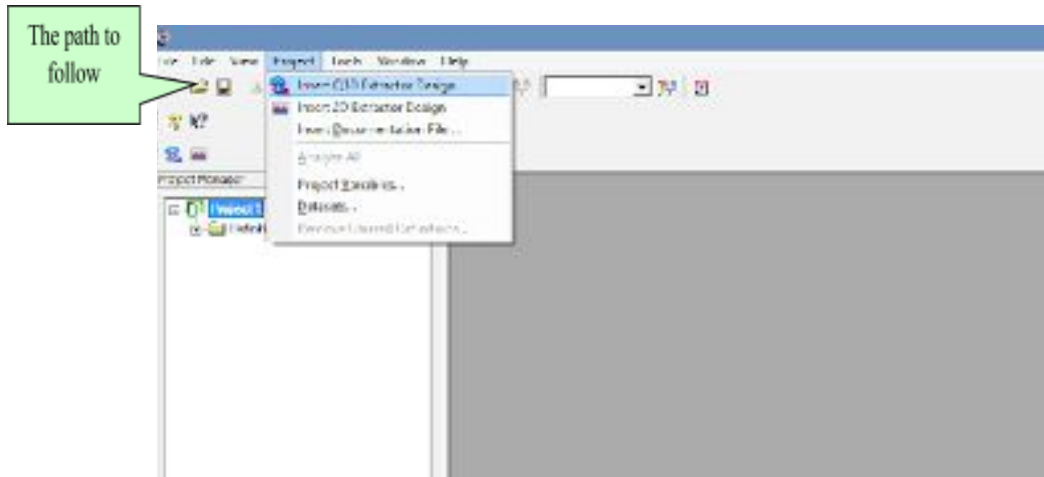
OK Cancel

Select All  
Layers.

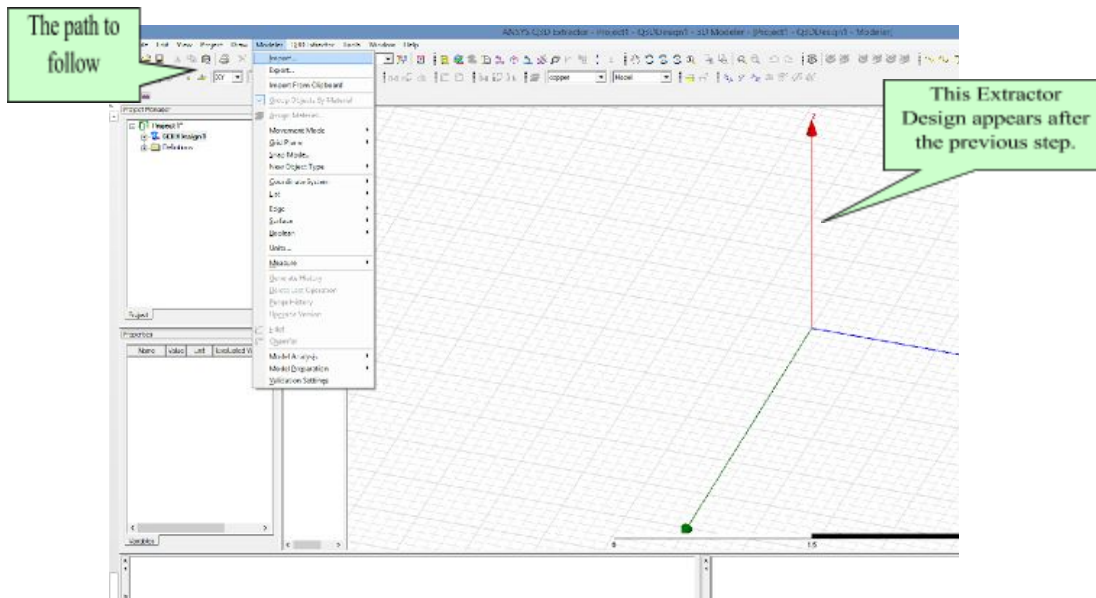
**NOTE:** If you select a version that is later than 2010, the file will not import into Q3D.

## Section 4: Importing DXF File into Q3D

1. Open Q3D
2. Click on *Project > Insert Q3D Extractor Design*.



3. Click on *Modeler > Import*.



4. Navigate to the DXF file you exported from Altium Designer and Open that file.

5. Place a check next to everything except the overlays, the solders, the mechanical layers, and the multilayer, and click OK.

## Section 5: Creating the 3D Model

### Subsection 1: Changing the Units

When the model is imported into Q3D, its units are in mm. For the purposes of the analysis, we need to change it to mils:

1. Click on *Modeler > Units*.
2. Select mils in the drop-down menu.
3. Check ‘Rescale to New Units’ and click *Done*.

### Subsection 2: Editing the Layers

#### Changing Materials:

Most of the layers and parts in the model are made of copper. However, there are certain layers, such as the Core and the Pre-preg layers that are made of FR4\_Epoxy. All parts of the model import in without an assigned material. To assign the material for an object:

1. Select all the objects you want to assign as a particular material, right click and click *Properties*.
2. Under Material, select Copper, and click *Done*.

#### Scaling and Moving the Board:

When the model is imported, the different layers and parts are superimposed on one another. Because of that, we must appropriately scale and move the different objects. Given below the scaling and moving sub-sections are the layouts for 2-layer and 4-layer boards. When you scale and move the layers of your board, you will have to refer to the layouts given after these two sections.

#### Scaling:

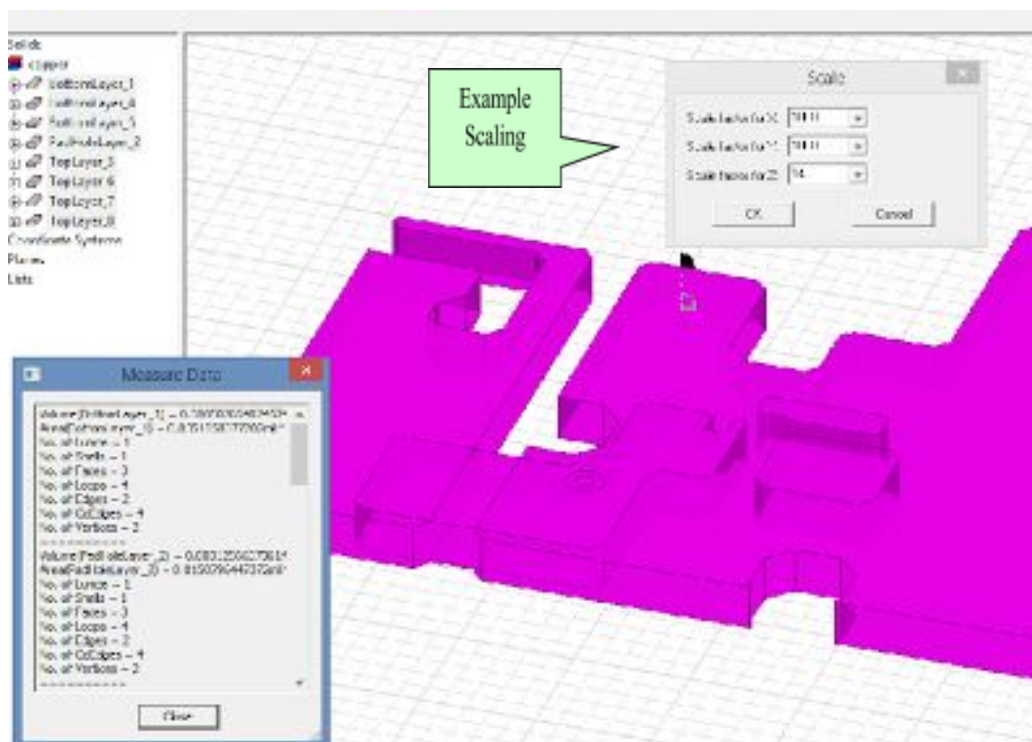
1. Open your PCB layout in Altium Designer.

**NOTE:** There will be two layers: the top paste layer, and the bottom paste layer, in addition to the top layer and the bottom layer. These layers go above the top layer and below the bottom layer resp. The thickness of these layers is the same as the thickness of the top and bottom layer. When assigning Sources and Sinks, as described in a following section, these will be assigned to objects in the paste layer.

**NOTE:** Use CTRL+M as a measuring tool in Altium.

2. Measure the edges of your layout and make a note of them.
3. Go back to your Q3D design.
4. Right-click and click on *Select Edges*.
5. Right-click and click on *Measure > Edge*.
6. Measure the edges of the imported layout and compare with the measurements from the original PCB layout.
7. To change the measurements of the import to reflect the PCB layout, select the objects you want to scale.
8. Right click and then select *Edit > Scale*.

**NOTE: For the thickness of the layers, refer to the layouts given below.**

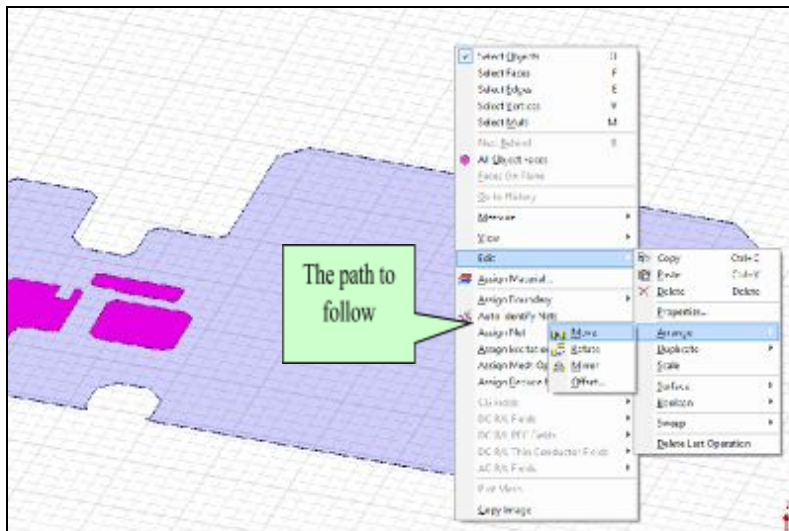


9. Change the X, Y, and Z scales.

## **Moving:**

Usually Q3D will name the parts in the different layers in a procedural manner, ie: TopLayer\_\*\*\* where the asterisks are a unique number for the layer. As a quality of life improvement, select all of the parts that belong to a particular layer, and right click > group > create. This will create a group of the parts that you can select to refer to all of the parts in a layer at one time. This will make it a lot easier to create the model correctly, and make it a lot less likely that you will accidentally misalign a piece in model reassembly. You will also be able to select a layer and *right click* > *view* > *show only selection* to view only the layer(s) that are most useful for your editing.

1. Select the group or objects you want to move.
2. Right-click and select *Edit > Arrange > Move*. (or click the move tool in the toolbar at the top)



3. Click on the object again.
4. Press Tab to take you down to where you can input how much you want to move the object in the X, Y, and Z directions.
5. Input the number you want, and hit Enter.

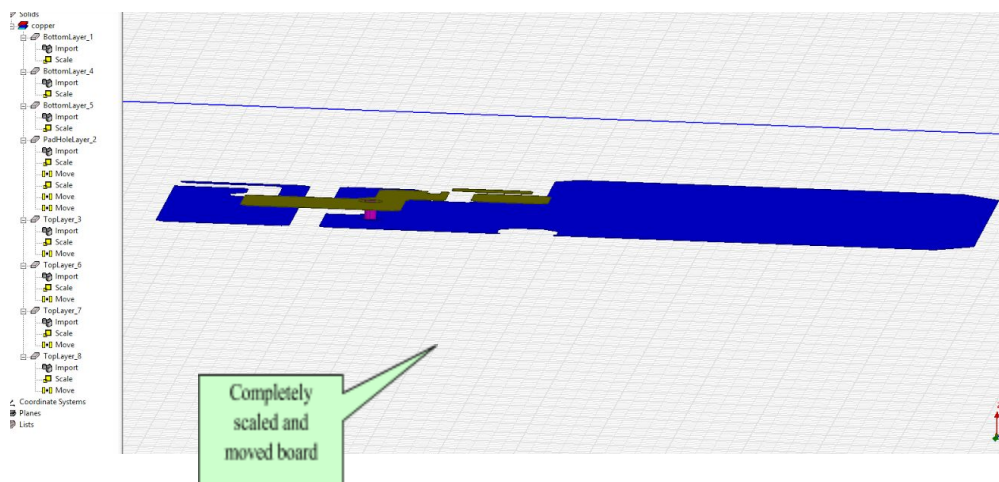
**NOTE: For the position of the layers, and the distance separating them, refer to the layouts given below.**

Enter  
numbers  
here



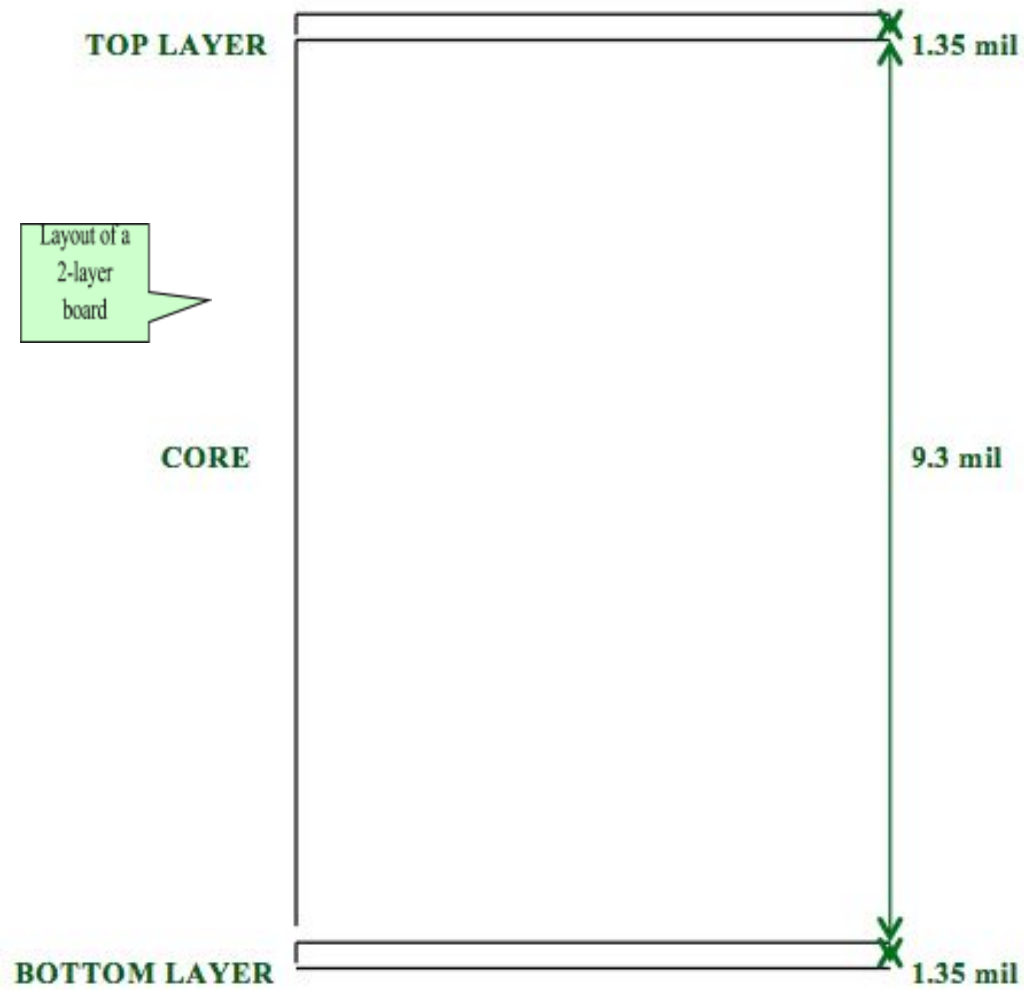


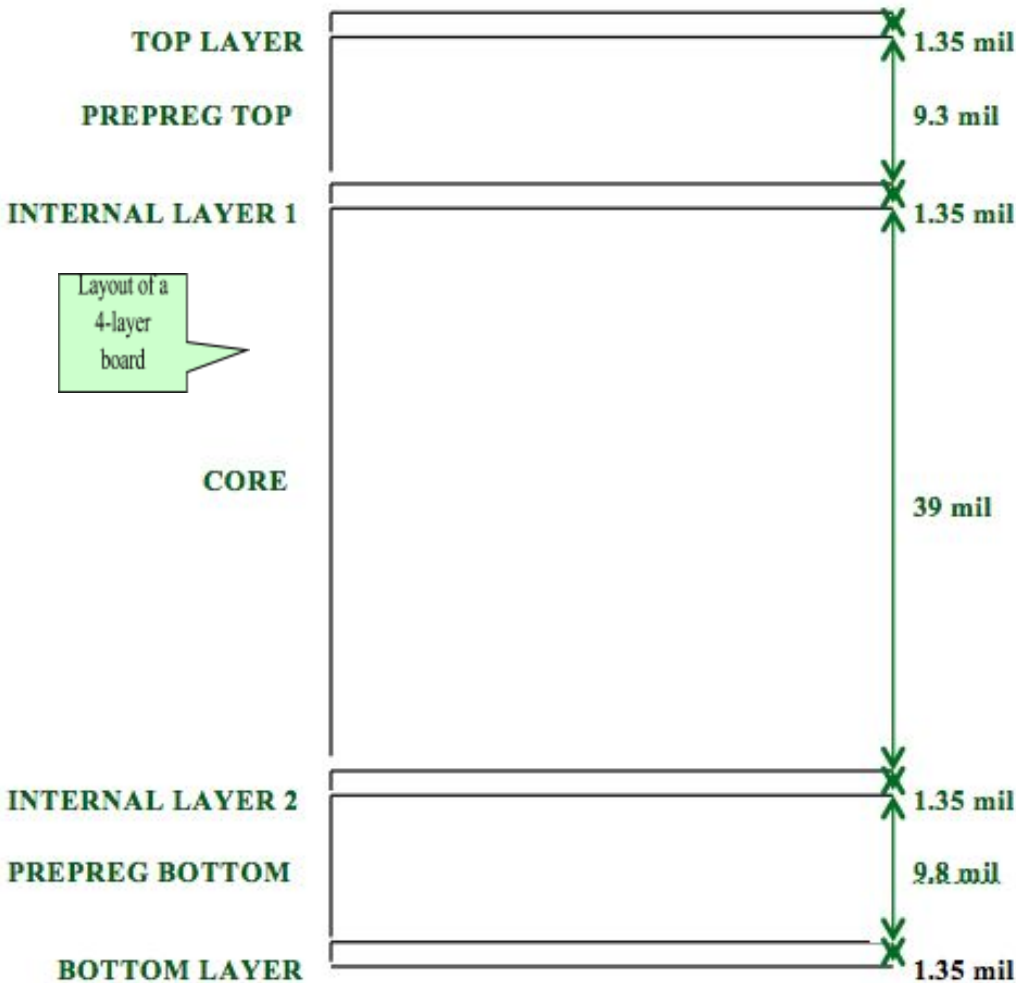
- a. After selecting the object(s) you can draw a vector in the model window that will move the objects by that vector instead. Depending on your model and what you are trying to do this may or may not save you time calculating the distances to move.
6. Continue until all the objects are where you want.



Given below are the two layouts for 2-layer and 4-layer boards as a reference for the space that should be left between the layers in your board:







### Subsection 3: Altering Unnecessary Parts

Almost all of the editing that will need to be done will be done through boolean operations within Q3D. The modelling system is based on using primitive or existing objects to edit other touching or overlapping objects to create complex models useful to simulation. All Boolean operations are performed in basically the same way.

1. Select all the objects you want to unite.
2. Click on *Modeler (or right click on an object) > Boolean > \*Operation\**.

The boolean operations that are most useful to the editing process for PCBs is Unite and Subtract. Uniting two overlapping objects will delete any overlapping

material between the two and removes any faces between them, making them one object. To understand subtraction, you need to understand how tool parts work within Q3D. When a subtraction is done there is a part that you are doing the subtraction, the blank part, and then another tool or victim part. Once you subtract, any overlap between the two parts will be removed from the volume of the main part, and typically the tool part is deleted. Most of the time when you are doing subtraction for the PCB traces, you do not want to get rid of the tool part, so make sure that the subtraction part is in the left box, and tool parts are in the right box, and that the “clone tool parts” box is checked at the bottom so that the tool parts will remain after the subtraction.

The main goal of this step is to make sure that:

- Each layer does not have any overlapping regions
  - overlapping regions will cause your model to not verify and not simulate
  - The import process will create many extra overlapping objects that together constitute the shapes you imported.
- Important pad/via locations are not removed
  - pad/via locations will also be represented by overlapping objects.
  - pad/vias must be modeled as separate objects to ensure that there is a face in the right place when excitations are defined later

To accomplish this you need to do two main things to your model. First identify all of the shape pieces that just make up the shape of your traces. Then unite all of those pieces so that you are left with traces with only pads that are overlapping. Then subtract all of the pads/vias from the traces they are a part of, and remember to clone tool parts. None of this need be done all at once, it is often easier to just work your way through a few pieces at a time. Note that holding control and clicking on the pieces you are interested in is often the simplest way to select the pieces you want.

At any point you can hit the validate button on the toolbar at the top. Everything will eventually need a check here, but for the modelling process this is the first time that the 3D model should validate. It may be a good idea to periodically check this to make sure you have not invalidated your model.

Now vias need to be added to your model if they were not included initially. This may vary somewhat depending on your particular geometry, however one of the easiest ways is to extrude the face of one of the placeholder disks to the plane it goes to. If you have done the last step correctly, you should have disks in your layers where the vias start and end. Look at the internal face of the disk, hit f to enable face selection mode, and click on the internal face to select it.

*right click > Edit > Surface > Create Object from Face*

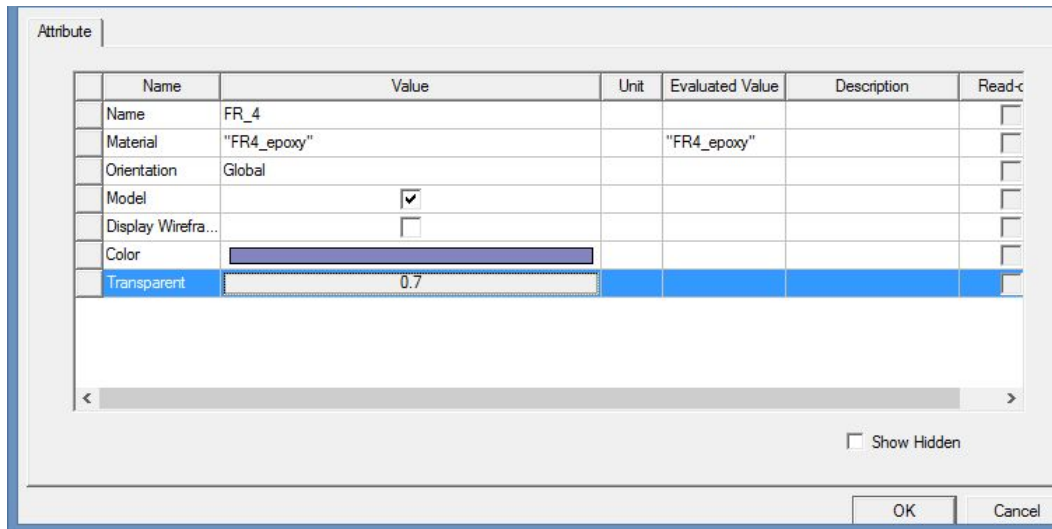
This will create a 2D sheet on the face of the disk where you want it to extend into a via. Select this new 2D sheet in your model panel on the left, and *right click > edit > sweep > along Vector* . This will now bring up the vector drawing panel. In the window get the selection point to snap to the middle or the side of the face, click, and then click again on the corresponding spot on the disk that the via should go to. If all goes correctly a cylinder should be created between the two disks that end the via.

It may be beneficial to unite the two end disks with the internal cylinder, to cut down on part count and save time avoiding the extra mesh that would be created in the disks. Once you have created all of the vias that will be necessary for the PCB that you are modelling, it is a good idea to check the 3D model validation to make sure that you still have a valid model.

#### **Subsection 4: Adding the FR\_4 Layers**

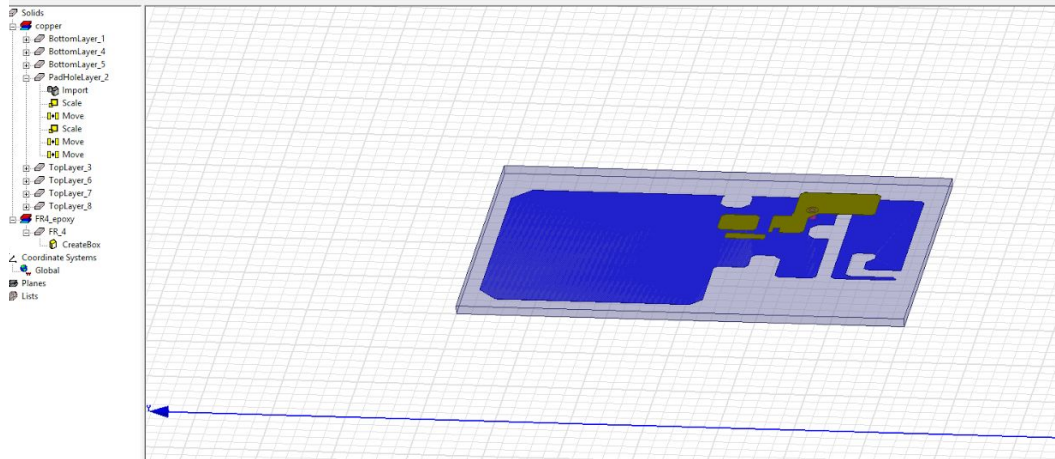
1. Create the middle layers (the Core and Prepreg layers) of the board using the *Draw > Box* command. To begin with you can create the box with arbitrary sides since you will be immediately changing them to the correct dimensions. Depending on your board, you may find that you want to draw in the layers separately, or as one large box. If the layers are all the same material, It should not matter as Q3D will treat the boundaries between adjacent objects the same. Doing separate objects will cause there to be more tetrahedrons created in the meshing process, so this may slow down your initial solution passes.
2. Right-click on the box to change its properties: assign the material to be FR\_4 epoxy, and change the dimensions to completely cover the space around the layers. *right click > edit > properties* will bring up the menu that will allow you to change the material of the box. You may need to scroll through the list to find FR\_4 or the material of your choice. To change the size of the box you have created find your box object in your model object window, expand it by clicking the '+' box, and double click

on CreateBox to open the window to edit the size and position of the box.



- a. Because of the way that the trace geometry is imported from your file, there can be complications in the meshing step when you try to simulate later on. It is possible for a model to verify, but then fail meshing once the simulation begins, under certain circumstances based on the imported geometry. By making the FR4 layer completely engulf the traces on all sides (ie: make the FR4 layer ~1 mil larger in all dimensions than the edges of the traces), you can sometimes avoid some of these meshing mistakes. This may not be appropriate for all geometries and designs, but may also avoid complicated errors for cases where it is less important. If you are have a validated model, and are getting errors and this is not appropriate/does not solve the problem when you analyse, you need to manually address the problem. This is more complicated than the scope of this guide but you can start by clicking *Modeler>Model Analysis>Show Analysis Dialog> Last Simulation Mesh* to see the individual meshing errors from the last simulation attempt.
3. In this layout, the via and the FR\_4 layer occupy the same space. So, we must subtract one from the other. This is similar to the subtraction

operations that were done when you were healing the imported model.

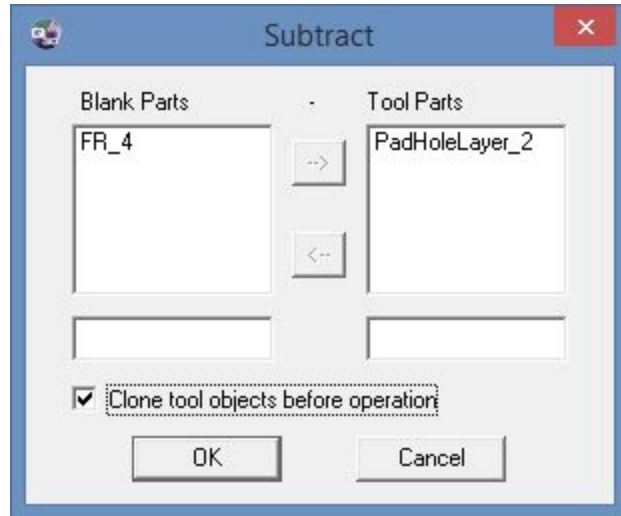


4. Click on FR\_4 and the via. As long as your model is not ridiculously complex, there is not really a downside to selecting all objects for the subtraction operation at the same time. Doing this will ensure that you don't forget about anything that may be overlapping with the dielectric layer you just created.
5. Click on *Modeler > Boolean > Subtract*.



**NOTE:** If the Clone box is not checked, you will end up completely deleting a layer, or object. So, it is very important to check it off.

6. Verify that FR\_4 is under Blank Parts, and that the via is under Tool Parts.
7. Check *Clone tool objects before operation*.



8. Click *OK*.

### Subsection 5: Validating the 3D Model

1. Click on *Q3D Extractor > Validation Check*.
2. Verify that you have a green check mark against 3D Model.

**NOTE:** If you have a red cross against 3D Model, the Comments section in the bottom left of the screen will outline the errors in the Model. This will help you fix your errors.

## Section 6: Nets and Excitations

A Net is a collection of touching conductor objects separated by non-conducting materials or by the background material.

Sources and Sinks are excitations assigned to each net, which are sometimes needed to generate solutions for the analysis of parasitics. To generate a capacitance solution, you do not need to assign any Sources or Sinks. However, for resistive or inductive solutions, any net that you want to include in the matrix solution must have at least one Source defined, and a single Sink. Attached to each Source is an independent current source. The Sink collects all the current injected at the Sources and allows it to flow out of the conductor and back into the independent current sources, completing the electrical circuit. Imagine that the current enters the trace at the Source and leaves at the Sink.

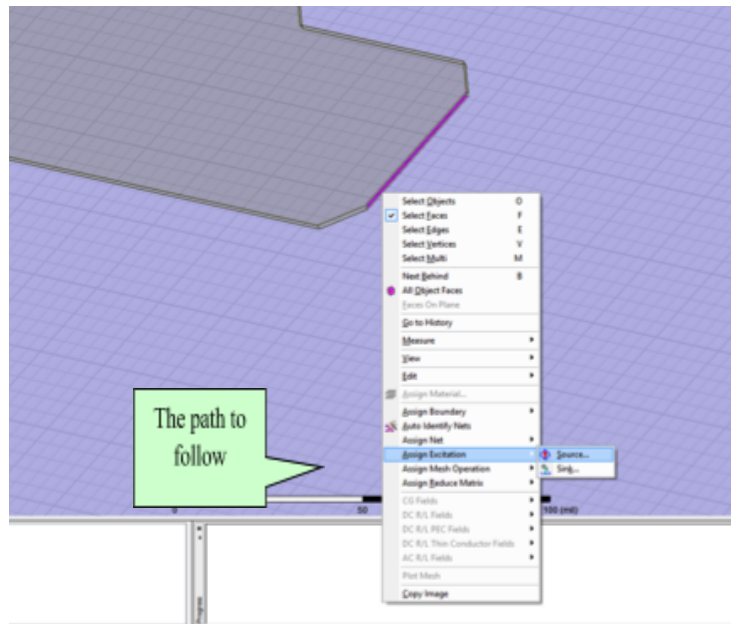
### Subsection 1: Assigning a Source

1. Right-click and click on *Select Faces*.
2. Select the face you want to assign as a source.
3. Right-click and select *Assign Excitation*.
4. Select *Source*.

**NOTE: Nets can only be assigned to conductive materials. So, remember to change the material of the imported traces to copper!**

**NOTE: A whole face of a 3D object must be chosen to apply an excitation.**

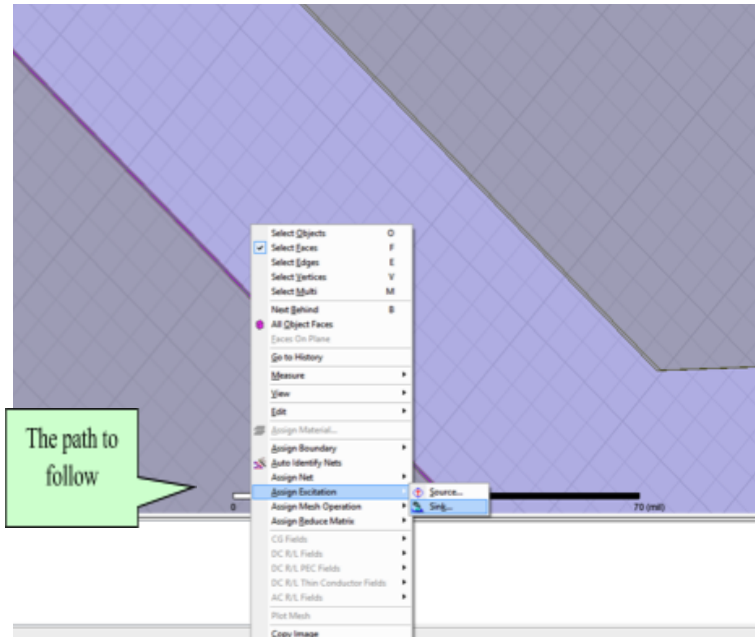




5. Rename the Source if you wish to, and then click OK.

## Subsection 2: Assigning a Sink

1. Right-click and click on *Select Faces* (if you have not already done so).
2. Select the face you want to assign as a sink.
3. Right-click and select *Assign Excitation*.
4. Select *Sink*.



5. Rename the Sink if you wish to, and then click OK.

### Subsection 3: Identifying Nets

If you have two conducting objects touching each other, Q3D treats them as one combined object. Q3D can auto identify nets by finding these touching conductors.

1. Right-click and select *Auto-Identify Nets*

**NOTE: If there are existing Nets in your design, some assignment information may be changed by Auto-Identifying.**

**NOTE: You cannot assign Nets to unconnected conducting objects.**

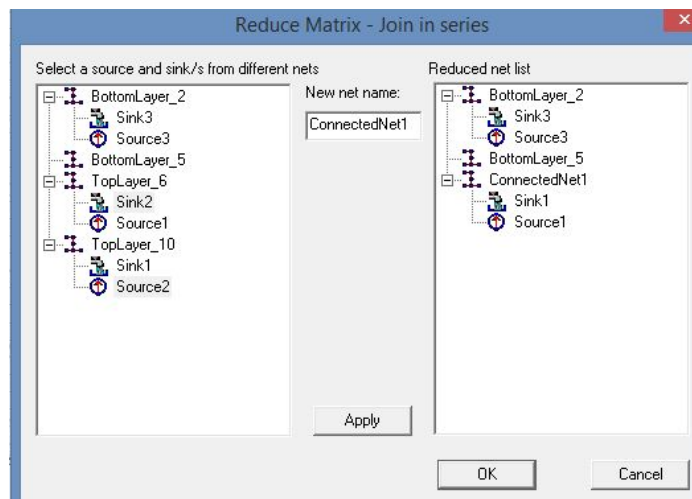
## Section 7: Reducing Matrices

When you generate a solution in Q3D, a row and column is generated for each Net in the capacitance matrix, and a row and column is generated for each Source in the resistive and inductive matrices. You can modify these matrices to generate different solutions by using the Reduced Matrix option.

For our analysis, we will be measuring the inductance in a loop of traces. The Join in Series option for reduction allow you to connect the Sink terminals of one or more conductors (or Nets) to the Source terminal of another conductor. Then, these conductors are treated as a single Net. There are also certain traces where we will have two or more Sources that contribute equally to the current in the trace. The Join in Parallel option for reduction allows you to connect these Sources in parallel. These Source terminals are then treated as a single object.

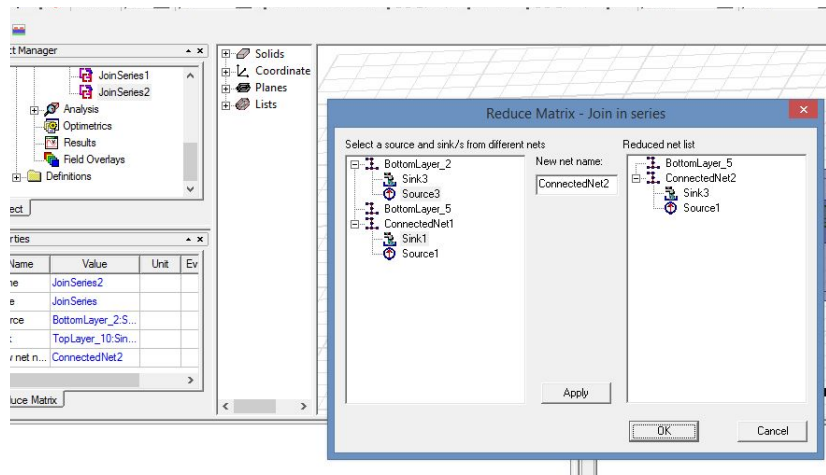
### Subsection 1: Join in Series

1. Expand *Reduced Matrix* in the Project Manager Window.
2. Right-click on *Original* and select *Join In Series*.
3. Select a Sink terminal from one Net and a Source terminal from another, to connect these Nets together.



4. Enter a name in the New Net Name box.
5. Click *Apply*, and then click OK.

6. Repeat steps 1-5 until the loop is formed. Here, you will have to select the Reduced Matrix you created instead of the Original matrix to add to the loop.



## Subsection 2: Join in Parallel

1. Expand *Reduced Matrix* in the Project Manager Window.
2. Right-click on the last Reduced Matrix you created and select *Join In Parallel*.
3. Select two or more Source terminals to connect in parallel.
4. Enter the New Net Name, the New Source Name, and the New Sink name.
5. Click *Apply*, and then click OK.

After you have performed all the operations you need, you can delete all the matrices except for the last one, and rename the last one into something of your choice.

## Section 8: Creating a Solution Setup

For the Solution Setup, you will need to specify the frequency at which you want to generate the solution. Q3D computes resistance and inductance solutions for both DC and AC problems. Depending on the frequency of the analysis, either the DC or AC solution is valid, and you will need to determine which of the two is valid for your frequency to obtain accurate results.

In the DC frequency region, both resistance and inductance are nearly constant with frequency. In the AC frequency region, the inductance is nearly constant with frequency, while the resistance increases in proportion to the square root of the frequency. The AC self-inductance is lower than the DC value because of skin effects. However, these effects modify the skin depth of the conductor, reducing the effecting cross section of current flow, and increasing the resistance.

### Subsection 1: AC or DC region

1. Determine the region your solution frequency is in.

The range of frequencies that are in the AC region is given by the following formula:

$$f \geq \frac{9}{\pi \sigma d^2 \mu_0 \mu_r}$$

Here,

- $\sigma$  is the conductor's conductivity in S/m.
- $d$  is the thickness of the conductor in m.
- $\mu_0$  is the permeability of free space, which is  $4\pi \times 10^{-7}$  Wb/Am
- $\mu_r$  is the conductor's relative permeability.

**NOTE: The thickness of the conductor is 1.35 mil for our**

**traces. NOTE:  $\mu_r \cong 1$  for most metals.**

The range of frequencies that are in the AC region is given by the following formula:

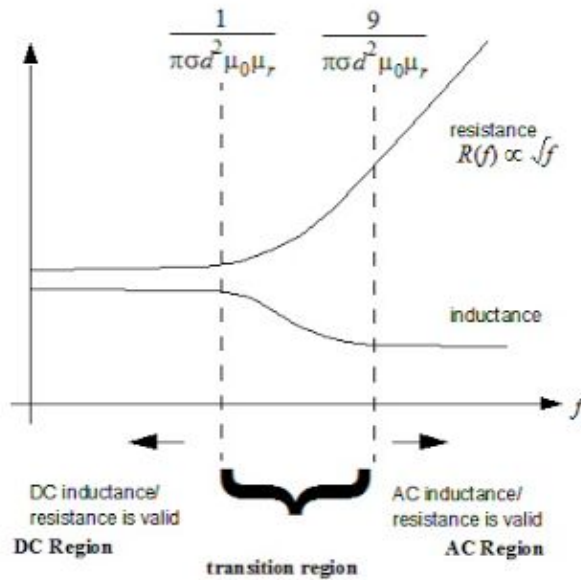
$$f \leq \frac{1}{\pi \sigma d^2 \mu_0 \mu_r}$$

Here,

- $\sigma$  is the conductor's conductivity in S/m.
- $d$  is the thickness of the conductor in m.

- $\mu_0$  is the permeability of free space, which is  $4\pi \times 10^{-7}$  Wb/Am
- $\mu_r$  is the conductor's relative permeability.

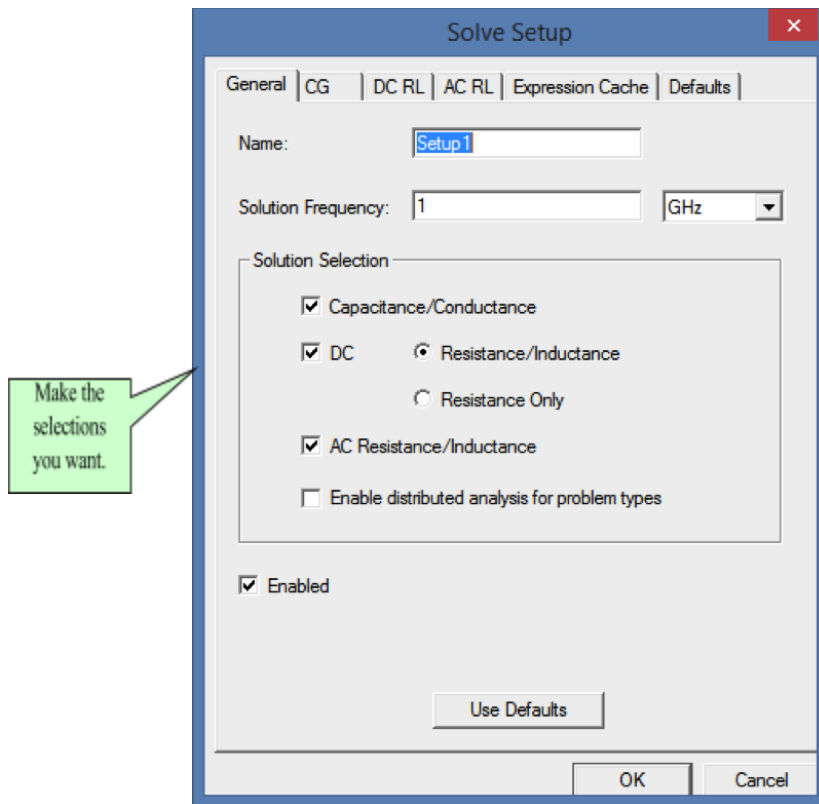
The different frequency regions:



After you have determined which solution, AC or DC, is valid for your solution frequency, you can create the Solution Setup.

## Subsection 2: Creating a Solution Setup

1. Right-click on *Analysis* and select *Add Solution Setup*.
2. Leave the default name of the Setup, or enter a new name.
3. Enter the solution frequency in the appropriate box.
4. Place a check next to the solutions you wish to find. Depending on your solution frequency, and the corresponding region, select either AC or DC in the Solution Selection box.



5. Click OK.

## **Section 9: Analysis Considerations and Meshing Operations**

When Q3D simulates your model in the adaptive pass mode, as it usually will, the first step is to create a mesh for your model. It is somewhat of a complex process, but basically the software will break your model down into tetrahedral pieces that it solves for fields within and at the boundaries with neighboring tetrahedrons. Q3D will create an initial mesh, and then for further passes, it will refine the mesh, breaking the most important tetrahedrons down further into multiple more tetrahedrons, to improve the accuracy of the solution further. Because it is the basis of the solution method, your mesh is integral to the accuracy and representation of your model in the final solution that Q3D creates. The meshing algorithm will naturally create a lot of tetrahedrons in small, intricate pieces, as well as on curved surfaces, so that the mesh will correctly represent the model.

This can be good or bad, depending on the situation. If too fine a mesh is created in an unimportant place, then a lot of solution time will be added while the program solves for values in relatively insignificant places. If the mesh is too coarse then it can potentially be inaccurate and/or miss essential geometry features.

The most general way that you can work with these mesh operations is to just specify the fineness to fit your model and the results that you are trying to get. To do this go to the Mesh Operations icon in the project manager *right click > Initial Mesh Settings* . There are a lot of specific settings you can tweak if you go into the manual mode, but just using the slider provided works pretty well for the majority of cases.



The most common place that your mesh will be too fine is on curves on your traces or on your vias due to the meshing software trying to accurately represent the curves. Because it will try to accurately represent the curves by default very accurately on the first mesh approximation, the vias and other curves may be overrepresented in the mesh that is created first. This will cause the simulation to be slower on all passes and make it take longer to converge. Often the perfectly circular nature of the geometry is not as important as the compromise in performance. One way you could deal with this is to change the geometry to be more simple not true surfaces, but you can often get around this by doing equivalent approximations using specific mesh operations for those vias/curves that are troublesome. If the suspect regions turn out to be important to the current flow and result accuracy then they will be refined in subsequent passes anyway.

First select all of the vias/objects that you want to make coarser. Then go the mesh operations tab in the project manager. *right click > assign > surface approximation*. This will bring up a similar slider to the one that you would see under the initial mesh settings. Drag this far to the leftmost few notches to apply this to the selected objects, and click apply. This can save a lot of time both in simulation and/or in trying to reshape an unwieldy imported geometry.

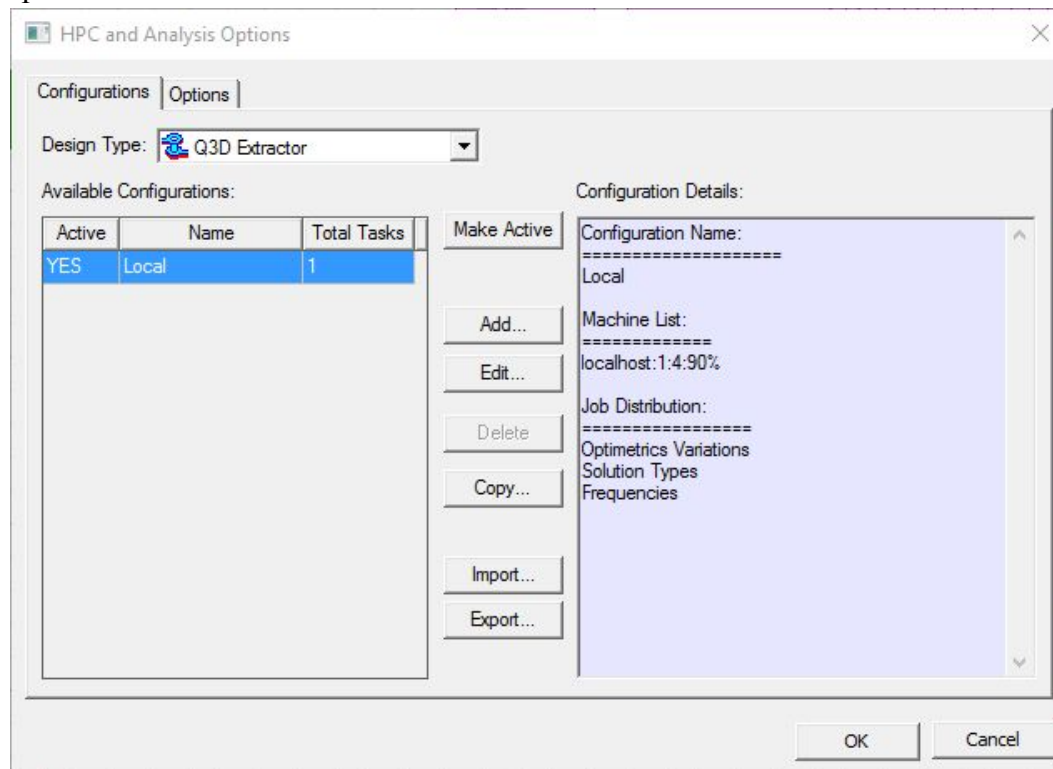
## Section 10: High Performance Computing Options and Setup

For many industry applications it is assumed that you will be running very large simulations on clusters of computers or at least very fast and expensive machines. For all of these cases high performance computing options need to be defined to be able to take advantage of the hardware at hand. By default ansys products only simulate using one core of the machine you are using. On fairly normal machines like the ones in most labs at miami that have Q3D this can be extremely limiting if you do not know how to change the setting to take advantage of what resources you do have. Running Q3D on only one core of a machine will lead to very long simulation times and low resource utilization at best.

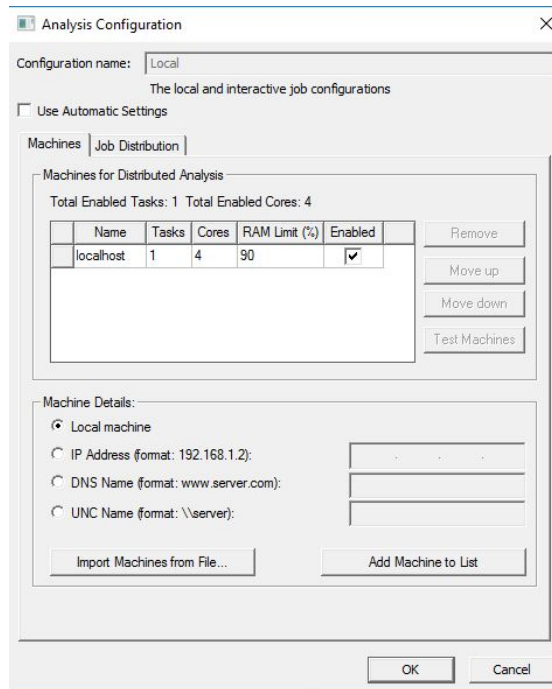
For most computers here at miami all we can do in the high performance options is to allow Q3D to be able to use all four (usually) cores of the machine we are on. To do this first click on the High Performance Computing and Analysis options button on the top of the toolbar.



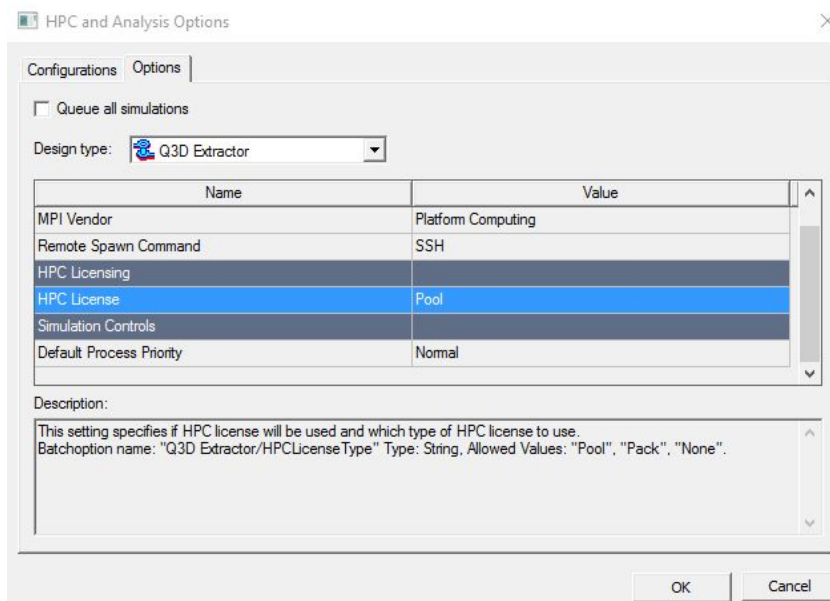
This will bring up the following box that you can then use to change the HPC options.



Hit Edit in the middle of the screen and change the field that says number of cores to four so that it looks like the following picture.



Hit 'OK' to return to the HPC menu and navigate to the options tab at the top of the screen. Under the HPC Licensing heading, change the type from what probably says 'None' by default to 'Pool' so that it looks like the picture below, and then click ok.



High performance computing options require separate ansys licenses to enable. The educational version of ansys that is in use at the time of writing includes the pool option that we need to enable multiple cores.

The Pack option is not a license that we currently have, but is what you would use if you were going to run the simulation on a cluster of computers. Cluster setup is outside the scope of this tutorial.

If at simulation run time, the program errors for a reason related to licensure, then the most likely cause is that your version of ansys does not have the required license (pool/pack) for the HPC settings that you have selected. Returning the core number to '1' and the license type to 'None' should allow you to run your simulation, albeit slowly.

## Section 11: Validation and Analysis

Before you perform the analysis, you will need to validate the Setup. To do this:

1. Click on *Q3d Extractor > Validation Check*.
2. Verify that you have a green check mark against every operation. If you have a red cross or a yellow warning, fix any errors before continuing.
3. Under Analysis in the Project Manager window, right-click on *Setup 1*, then select *Analyze*.

If you are running a simulation for the first time on a new geometry, or with more taxing settings, it may be a good idea to keep an eye on the resource monitoring section of your task manager. The most common cause of performance issues that have been run into as of writing this is for the RAM utilization to exceed the cap that is set for Q3D. You can change this RAM cap in the HPC options if it is necessary, but the default 90% is pretty good. If the Q3D simulation exceeds the RAM cap, you will be able to see on the resource monitor. If this happens then the simulation will begin to page virtual memory on your hard drive, your drive utilization will likely rise to 100% and your computer will appear to grind to a halt. Technically if you leave the simulation long enough it will finish, but if this happens it is definitely the best course of action to stop it and change some settings and try again.

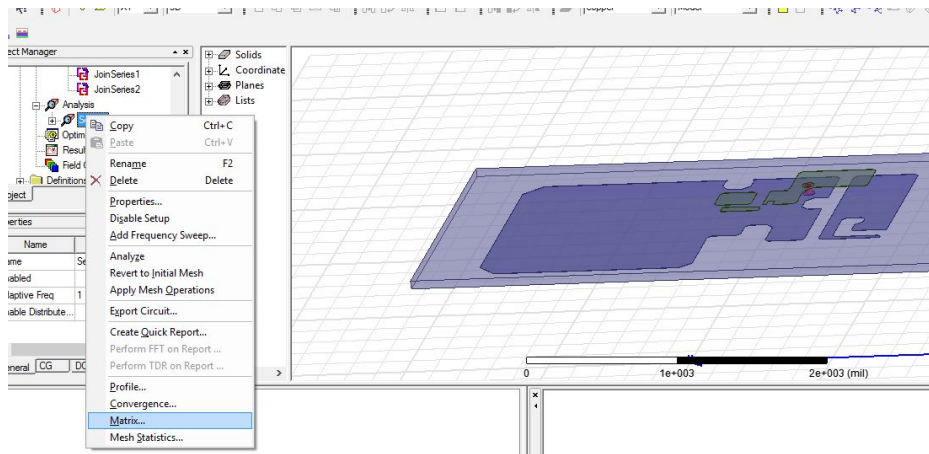
The most common cause of this is that mesh creation creates too many tetrahedrons for your machine to handle. There are a couple things that you can try to remedy this.

- If the simulation crashes after a couple passes in a simulation, but was fine before, you can likely leave your settings as they were, and reduce the number of passes, or the percent refinement per pass, or increase the goal percent error in the analysis setup. This will reduce the maximum number of tetrahedrons that the simulation will produce after refining the mesh each pass.
- Use mesh operations to either deprioritize less important areas of your model, or change your initial mesh settings to be more coarse in general. Both of these will lower the resolution of the mesh and possibly the result, but may lower the tetrahedron count enough for the simulation to run.
- If none of the above work, then you may need to simulate a less complicated model. Either strip away small and unimportant parts, or change complex geometries into more simple representations. The more simple the design, the less regions will be created so you may be able to then run the simulation.

**NOTE: A description of all the errors can be found in the Message Manager window at the bottom left.**

## Section 12: Results

1. Under Analysis in the Project Manager window, right-click on *Setup 1*, then select *Matrix*.



2. Select DC or AC analysis.

3. Select your Reduced Matrix. The inductance is the second number reported in the units selected in the inductance units box. If more than one cell shows up here then either your initial source/sink placement has extra paths, or your reduce matrix does not reduce all of the sources/sinks to one path.

