Board Station Tutorial

Written by Theodore Zeeff
University of Missouri-Rolla

Last Modified: April 5, 1998.

Purpose:

This tutorial describes how to design and manufacture a Printed Circuit Board (PCB) using Mentor Graphics.

Board Station consists of 5 distinct programs

? **Design Architect (DA)**
? **Librarian**
? **Package**
? **Layout**
? **Fablink**

The first of such programs, **Design Architect (DA)** will allow the user to conceptually draw out the design on the computer. Using circuit components such as basic logic elements, wires, buses, discrete components, etc., the design will quickly take form. Once completed, the design should look like one typically used by engineers for fine tuning and redesign.

To take this design information and make it into an actual circuit, you will need to specify every detail about the PCB and any part that goes on it. The program you will use to describe these properties is **Librarian**. Once all of this information is provided and saved, the design can then be taken to the next stage of development.

The next step uses **Package**. This program takes all of the information given by it in **Design Architect** and **LIbrarian** and links the two together. It also double checks anything you made yourself (parts, references to those parts...). Any errors made in the previous two steps will be found inside **Package.**

The next step involves **Layout**. This program takes the information generated in previous steps and lets the user layout out the PCB with it. Final construction of the PCB takes place here, and by the end of this program every detail about the board will have been specified.

After **Layout**, the desired PCB can be processed by **Fablink**. In **Fablink** , the design will be translated into a language the milling machines can interpret. These files can then be uploaded to the machine for fabrication.

Tutorial Sections:

Designing a PCB using Board Station:

? Back to: EE Home page

Board Station Tutorial

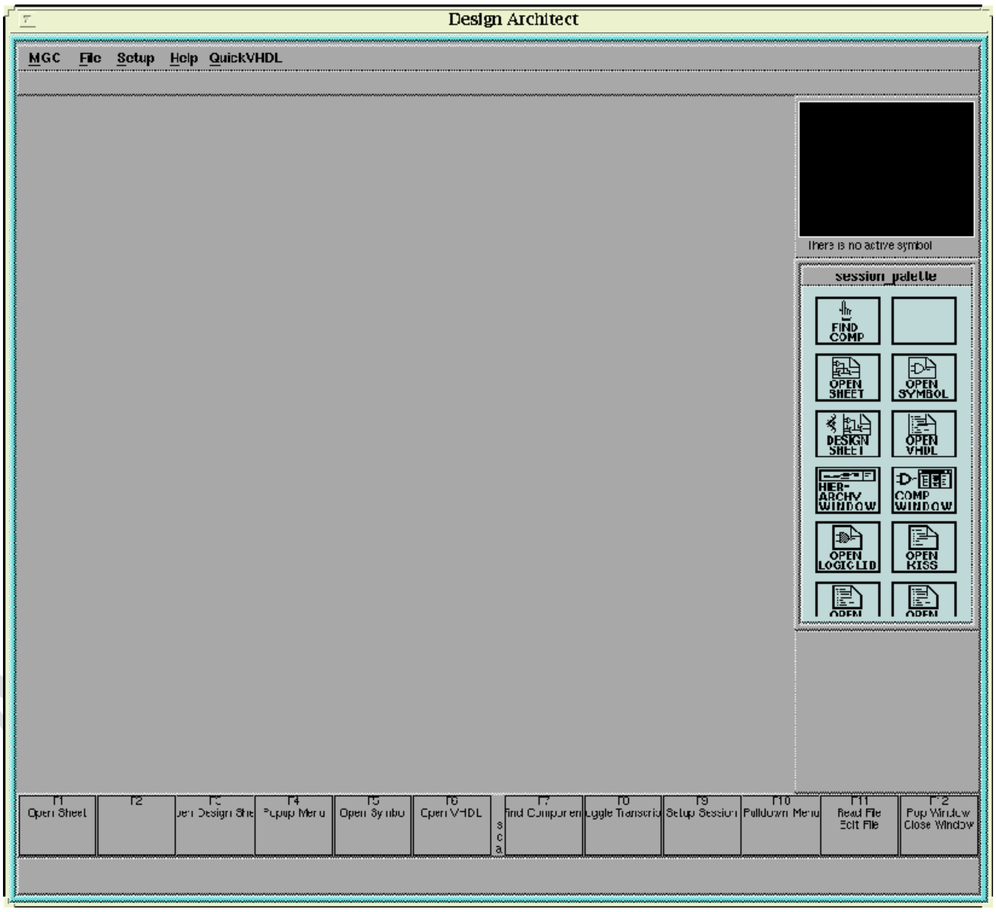
Part 3/7

Design Architect

Before you begin, make sure you are in the PCB directory and that 'sul' and 'swd' have been executed inside this directory.

To begin Design Architect, type **da &** at the eeclcxx% prompt. It will take some time to load the program. After about 20 seconds several disclaimers will have appeared and a window will come up on the screen. The screen should look something like the picture at the link below.

? Picture of Design Architect



To start a new design sheet, press **F1** or you may press the RMB (with the mouse cursor inside the DA window) and select Open Sheet from the menu titled Session (**RMB>Session>Open Sheet**).

Another window will appear titled 'Open Sheet'. For the component name, type in **Design1** or another appropriate name where $USERLIB was. Then click on OK. This schematic will then be placed in the working directory named PCB. All of the information pertaining to the design will now be placed inside the directory '/PCB/Design1'. **The name you give your 'sheet' should reflect what your design is. This name will be used to launch other programs in Mentor Graphics, so make it a good one.**

After a few seconds, you will see four different sections in the DA window.

? The first and largest will be the **Design Schematic** area. This particular window has the title :" Schematic #1" then the working directory: "PCB | Design1". This is where the schematic will be constructed.

? The second, which is in the upper right corner, is the **Active Symbol** window. This window displays the active symbol and what it will look like once placed in the **Design Schematic** window.

? The third area is the **Schematic Pallet** window, which is located in the middle far right portion of the Design Architect window. A variety of menus will be displayed here depending on what you are doing in the **Design Schematic** window.

? The forth area, located in the lower right corner of the Design Architect window, is called the **Context Window**. This shows the area presently displayed on the **Design Schematic** window relative to the entire design.

TIP:

There are additional commands for each window which can be accessed by clicking on the right mouse button (RMB) in each area's display window. Sometimes they can be used to speed up your design time. You may want to take the time to familiarize yourself with these menus.

Short Cuts:

Before we begin constructing the schematic, there are several shortcuts you should familiarize yourself with.

? The first involves using the middle mouse button (MMB). To use these shortcuts, just hold down the MMB in the Design Schematic window and move the mouse accordingly. A few of these are:

? **Enter:** short stroke to right.

? **ESC:** short stroke to left

? **Zoom OUT:** stroke to upper right

? **Zoom IN:** stroke to lower left

? **View All:** stroke to upper left

? **View Area:** stroke to lower right

? **Center About point:** two quick MMB clicks at desired location

? **Additional Help:** stroke in shape of question mark

? The second type of short cut uses the ones listed below the **Design Schematic** window. They are positioned in several different boxes, each representing a function key(F1,F2...) . The letters in the middle box represent SHIFT, CONTROL and ALT. To use a particular function hold down the appropriate 'shift', 'control' or 'alt' key and press a function key.

Making the Circuit Schematic

Currently there is only one reliable way of making a PCB at UMR, making each component by hand. One of the major disadvantages to doing this (aside from the extra work) is that the parts you make will not be fully simulated parts (although I believe it can be done...). There are Libraries of components available right now, but there are no geometry files that can be associated with them. Hopefully, by Fall Semester '98, there will be a complete set of simulation parts and associated geometries.

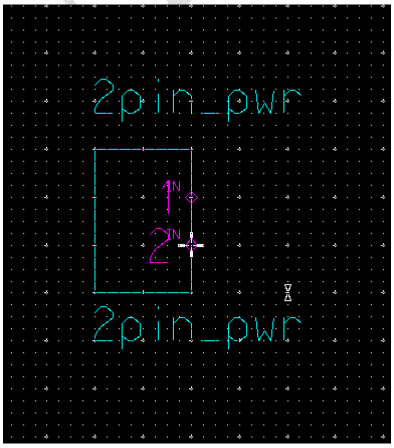
Making Symbols Not Found in On line Libraries:
The following is an example of a power connector. This part has two power connections.

To create the symbol that will represent your connector go to the pallet and choose **RMB>Display_Schematic_pallet**. Now click on the dark box containing 'Session'. Now select Open Symbol from the new display. A dialogue window will appear. Fill the window as follows:

Component Name: **2pin_pwr**
Options: **NO**

Another window will appear that looks similar to the one being used for your design. The name of the window should contain 2pin_conn somewhere in the title. The finished symbol will look like the link below.

? 2pin_pwr Symbol:



Will begin construction of the symbol by drawing the basic outline of the symbol that will define its boundaries in DA. Select **pallet>Symbol_Draw>Add_Polygon or Add_Polyline**. Now move your mouse cursor over the drawing area and click the LMB where you want the corners of your design to be. (Keep in mind that pins can only be attached to the large grid points displayed on this screen, so draw accordingly. Also keep in mind that you must end up at the same point where you started or you'll get an extra line on your symbol if you choose 'Add_Polygon'). An exact replica of the part is not needed at this point. Just draw a basic outline of what need to distinguish it from other parts on the DA schematic. Double click on a spot on the symbol when you are done.
Once the basic shape of the symbol has been made we need to add properties to it so that Package and Layout will be able to read it when needed. Make sure only the body of the symbol is selected and choose **RMB>Properties>ADD>Add_Single_Property**. Now click on **REF** in the window that appears. Now type in ' 2pin_pwr ' for the **Property Value:**. It is extremely important that the name of the property be exactly the same as the name of the symbol(keep in mind that it is case sensitive). Once you have filled in this information press 'ok'.
While still having the symbol body selected, add another property using the same method. This time select COMP as the property and again use ' **2pin_pwr** ' as the property value. This time, however, make the property hidden by selecting the 'hidden' option in the property window.

Now we will add pins to the symbol. Select **pallet>ADD_Pin**. Fill out the window as follows:
Pin Type: **IN**
Pin Names: **1,2**

The pin type is not critical to this symbol. The only difference between IN and OUT is what will be displayed near the pin in the DA circuit. This has no impact on the way the pin is read by any program.
The pin names should be written down if you were making a more complex symbol. The exact same names will be required by Librarian and a wrong name will lead to the pin, component and any trace attached to it not being read properly.

Now that the pin names have been made, you need to place the pins on the symbol. Click 'OK' on the 'Add_Pin' window. Now place the pins as necessary.

If you want to move the Pin Names, go to Setup in the menu bar. Now choose 'select filter'. When the new window appears, click on Set All. Now select 'OK' Now move your mouse cursor over the Pin Name you want to move. Now hold down the RMB (Right Mouse Button) and move the mouse cursor over the Move [a-MMB] selection. Now release the RMB. You should now be able to move the name of the pin anywhere you want to. Move the mouse till the name is in its desired location and press the LMB once.

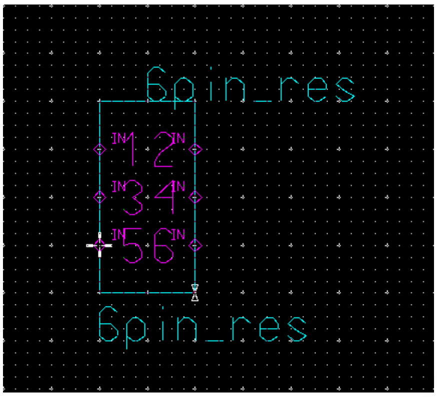
Once you are done making the symbol you will need to check it. Select

Check>With_Defaults. If it checks out properly then select **File>Save_Symbol**. After this is done you can close the symbol window without losing the new symbol.

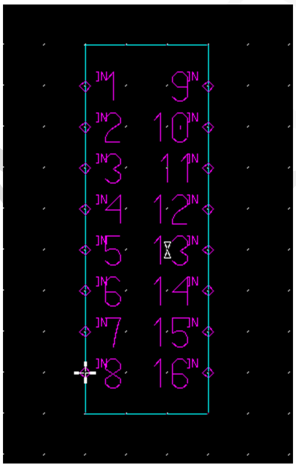
To use the symbol in DA first click the LMB anywhere on the Design Schematic window. Now from the pallet choose **pallet>Choose_symbol**. From the list of directories displayed in the window choose 2pin_pwr. Click 'ok' to close the window. Now you should see your symbol in the Active Symbol window. Place the part where necessary.

The other parts done as examples are the **6pin_res** and the **16pin.** The symbols are shown at the links below.

? 6pin_res Symbol:



? 16pin Symbol:



Selection/Deselection:

When you place parts in Mentor Graphics, the part just placed remains selected and active until you deselect it explicitly. If you have more than one part selected at a time, this may cause problems. For instance, if you want to delete a part that was placed wrong or didn't belong there and you press the 'del' key to delete that part while more than that part is selected, you will delete all active selected parts. To prevent this from happening, deselect the part by pressing 'F2' after the part has been placed. A quick way to find out how many parts are selected is located just below the main title bar. If you were to select the 74ls00 chip to place in the Design window you might see:
sel (+1) (w|dae) (Design1|schematic|sheet1)(74ls00|MG_STD.POS)()
The number after 'sel' in the parentheses is the number of selected items. After pressing 'F2' the number goes back to '0+'.

To select a part click on the body of the symbol. When the part is selected it will turn white. If you did not select the part, it will remain purple. You may also want to select a number of parts in a certain area. To do this you can hold down the LMB (near the part) and move the mouse around until the 'rubber band' encloses part of the symbol. Double check your selection by noting the number of selections after the 'sel'. Notice how the ends of the pins near the purple diamonds can be selected too. For moving and deleting purposes you should not worry about this feature. Just click on the body of the symbol and edit as necessary.

If you make a mistake, you can correct your last edit by selecting >Edit>Undo.

Wires:
After an appropriate number of logic symbols and misc. parts have been placed you will need to connect them with wires. To do so press 'F3' or select add wire from RMB>ADD>Wire: . A tool bar will appear at the bottom of the main window labeled 'add WI' (this will remain in place while you are making wires).
Start a wire by going to an appropriate place on the design and clicking the LMB. Now move the cursor to where it needs to be connected and press the LMB again. You can 'bend' the wire by pressing the LMB where appropriate. (If you need to branch the wire, simply continue connecting one branch and once completed, backtrack, with same wire, to the point where it needs to branch off). Keep in mind that you should try not to use two or more wires to make one circuit node. Just backtrack through one branch and split off where necessary. Once you have completed one node, use the MMB shortcut for 'enter' to complete the wiring. Don't forget to press 'F2' to deselect the wires.

If for some reason you need to connect two wires into one, select the pieces of wire that need to be connected, then use the command **RMB>Connections>connect_all**. Your wires are now connected. Remember,good wiring technique and consistent checking will keep you out of trouble later on. See the Checking Sheet section below for details on checking.

Adding Text to Design Schematic:

You may want to add comments to the schematic from time to time to clarify the design. To add such comments you need to select "text" on the schematic pallet on the right hand side of the screen, then "add comment" in the menu. Next a tool bar will appear with the label 'text'. Simply type your comment, press enter and move the text accordingly.

If you desire to change the height of the text then you need to change the selection filter so it can be selected. Go to **Schematic>text>set_selection_filter** and activate the comment text. Now select the text region with the LMB then press **RMB>Property/Text>Change_Height...** . Notice that you can do more than just change the height. You can also Move, Delete and Change Values (what text is displayed). Choose 'height' to change size of text. Note that the default text size is .1875 inches.

Simplifying Designs:

Globals:

Sometimes you will want to connect to wires, planes, or components together without having to connect them on the schematic with wires. In such a case, you will want to add a global bus to your schematic. Adding a global bus to your design will allow you to save time and effort into making your schematic look nice and possibly make your design easier to read.

To make a global bus, choose **Libraries>MGC_Libraries**. Now choose **Connectivity Symbols** from the palette menu. You will have to change the name of the global variable to get it to work properly. Two or more Global's with the same name will be connected, others will not be. Position your cursor over the 'G' near the global and press 'Shift+F7'. Type in the new name in the window provided.

Checking Sheet and Saving:

When your design is completed or you just want to save your design go to **Check>Sheet>With_Defaults**. The program will now check your design to see if you have made any serious mistakes. After it is done assessing your design, a comment window will appear. Inside the window will be displayed various accounts of the checking process. If any errors are present in your design, it will display them.

If you select text in the 'Check Sheet' window and switch back to the 'Schematic' window, the area on the schematic that the high-lighted text was referencing will be selected. This can be useful when checking complicated designs.

When you want to save the design, with or without errors, go to **File>Save_sheet**. When its done and you are ready to leave, click the RMB on the main DA window title bar and choose 'QUIT'.

? Next Part of Tutorial: Librarian
? Back to Main Page: Go to Beginning

Board Station Tutorial

Part 4/7

Librarian

This part of the tutorial will center around building individual parts needed to make the entire PCB. Such parts as the board itself, a multi-pin connector, through holes, pad stacks, component geometries and company logos. The information here should give you a head start towards building any geometry required of you.

To begin Librarian you must invoke it on your design. First make sure that the working directory is still the directory above your design directory by typing **sul** and **swd** in the PCB directory. Next type the following command: **librarian Design1** (Use the same name you used when opening a sheet in Design Architect).

Librarian should now load up on the computer. It will take around 20 seconds depending how fast the system server is working at the time. Once it has stopped loading you'll need to get rid of the report window by double clicking on the box in the upper left portion of the window.

For most designs you will need to create a default pad stack geometry which will be used when no pad stack is defined for an object. After this is done you should make the board itself and then all other geometries you need.

The following procedure details the process by which pad stack geometries are defined in Librarian. You may not need to make all three as listed (They are there for purpose of example) but you should make **at least one through hole pin** and **one via** so you have default geometries to fall back on in later steps.

Creating Pin Pad stacks

If you are using a part that doesn't come from the Board Process Libraries then you will need to create the part geometries and the geometry of the pad stacks that connect the part to the PCB. All of these geometries will then be stored in your account so that you will not need to build them again.

Through hole Pin

From the top menu select: **Geometries>Create_Geometry> Thruhole_Pin...**

Then fill in the dialog box that comes up as follows:

Pin Name: **t032rd**
Drill Size: **.032**
Units: **Inches**

No PAD Shape

SIGNAL Shapes
Layer: **Signal**
Circle
Diameter: .**060**

POWER Shapes
Layer: **Power**
Circle
Diameter: **.070**

(Solder Mask)Single Shape
Circle
Diameter: **.070**

No OTHER Shapes

When you are done, press OK. A new window will appear in the Librarian session. Observe and verify that it is what you want. If you don't like what you see then just close the window and don't save. Otherwise continue making the rest of the geometries or save the geometry following the procedure below for saving pad stacks.

The standard form for the name of the pad stack is as follows:
t032rd
t: This stands for through hole
032 : This stands for the diameter (in inches *1000) of the drill hole
rd : This stands for the basic shape of the pin, in this case, round.

Standard Drill Sizes:
.032
.040
.060
.080
.120

Other drill sizes larger than .080 can be made, however, try not to use them if you can. They require more work to make when you get to the milling machine portion of PCB creation. **Surface Pin**

From the top menu select the following: **Geometries>Create_Geometries>Surface_Pin**

Now fill in the box as follows:

Pin Name: **s70x30**
Units: **Inches**

(Pad)Single Shape
Rectangle
Width: **.070** Height: **.030**

(Solder Mask) Single Shape
Rectangle
Width: **.080** Height: **.040**

No Paste_Mask Shapes

No Other Shapes

When you are done, press OK. Note that the name of the pad stack comes from what type of dimensions the object has.

Through hole Via

A through hole via is a type of pad stack that allows the milling machine to drill down to another layer on the PCB and connect to that lower layer. To create a via through hole choose the following from the top menu: **Geometries>Create_Geometries>Thruhole_Via** and fill in the window as follows...

Via Name: **v032rd**
Drill Size: **.032**
Units: **Inches**

No Pad Shape

Signal Shapes
Layer: **Signal**
Circle
Diameter: **.050**

Power Shapes
Layer: **power**
Circle

Diameter: **.060**

(Solder Mask)Single Shape
Circle
Diameter: **.060**

No Other Shapes

Saving Pad stacks

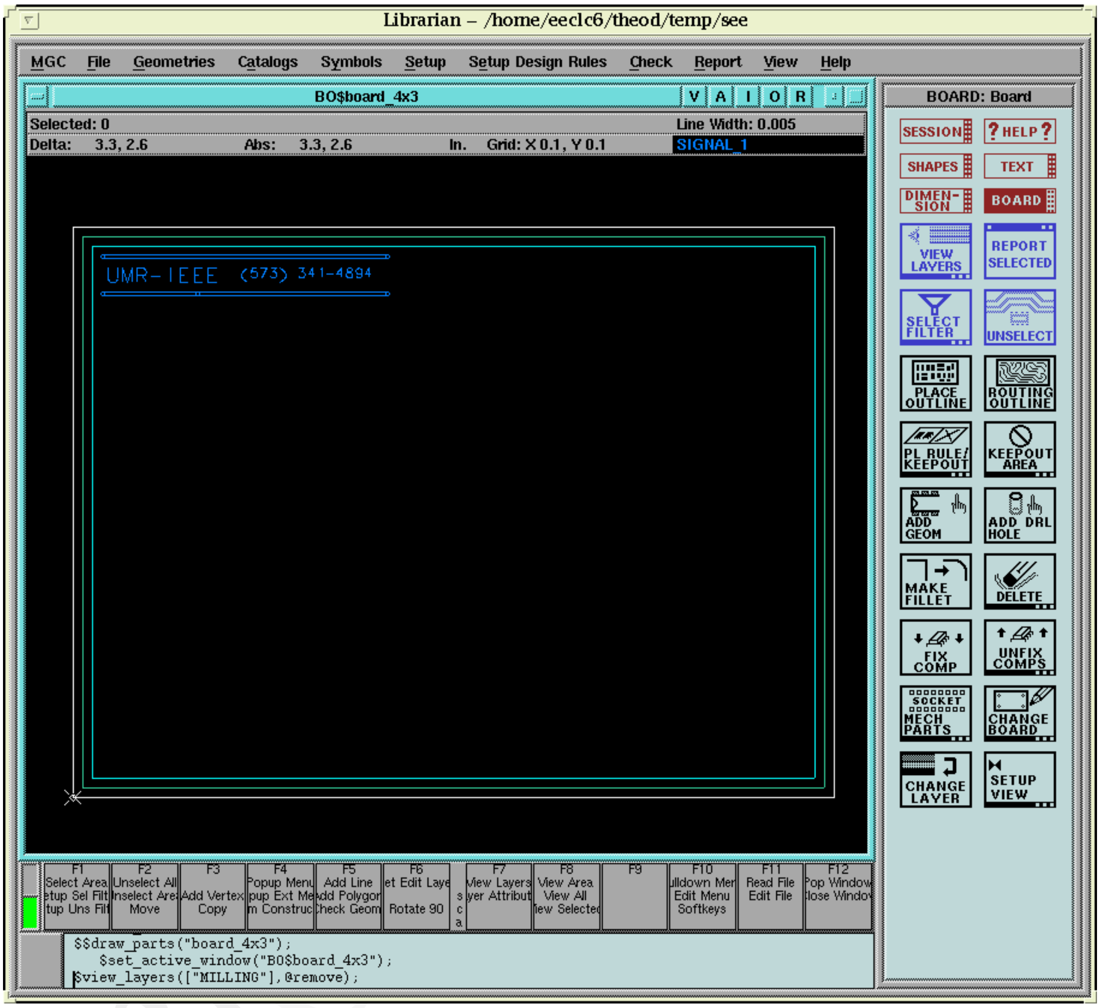
To save the geometries you have just created choose the following from the top menus: **File>Save>ASCII_Geometries...**
Then fill out the box in the following manner:

Geometries to save: **All Geometries (or Specific Geometries)**
Separate File for Each
Library to Store the Geometry: **Design**
Replace Existing File(s)

After you have done this, a dialog box will appear and confirm your saving them. Now you are ready to quit Librarian if you need to by clicking the RMB on the main title screen and choosing 'quit'. It will then ask you if you want to save your work. Choose 'yes' and if there are errors then you can still save if you need to.

Building a Board Geometry

? [Picture of Completed Board with Logo]



To make a board geometry do the following from the top menus: **Geometries>Create_Geometries>Board...** Then fill out the dialog box as follows:

Board Name: **board** (or some other appropriate name)

Default pad stack Name: **t032rd** (or choose one you made above)
Number of Routing Layers: **2** (or any number you wish)
Default pad stack Size: **.060** (may need to change depending on default)

Route Power Nets: **On Signal Layers**

Click on 'OK' to close the box.

Now you are ready to begin constructing the basic outline of the PCB. First zoom out using the shortcut keystrokes until the maximum vertical and maximum horizontal coordinates are above what you need for the dimensions of the board. Move the cursor around the screen to find out what they are. The coordinate system is below the main menu bar (use absolute coordinate numbers).

Now you need to setup the grid to which the cursor will 'snap'. Go to **Setup>Grid** and type in .1 next to 'X Increment'. The 'y' increment will default to the 'x' increment if you don't specify it. In this same window go to the 'display interval' and type in 2. Now press 'OK'.

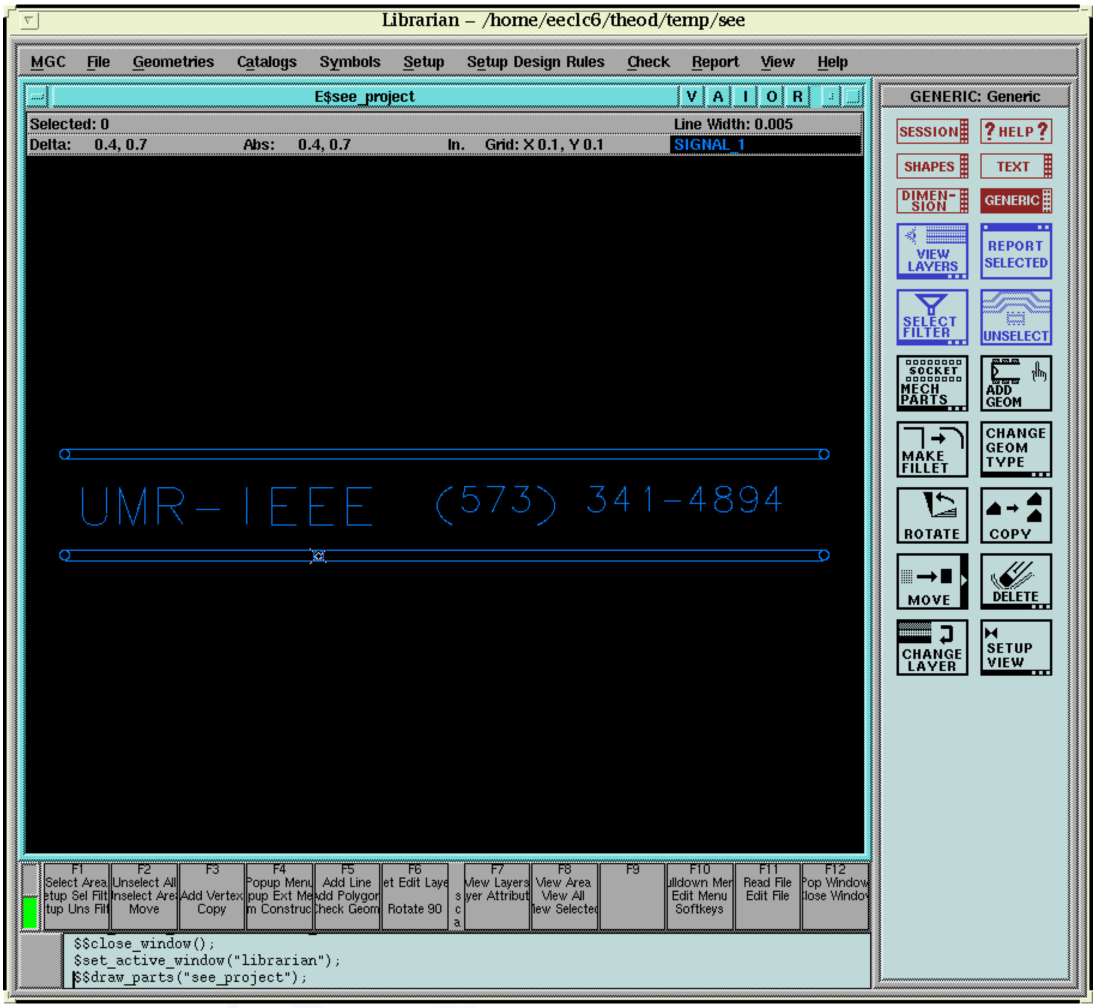
Start by selecting **RMB>(TOP MENU)>SHAPES>Add_Polygon**. Once you have done this you should see a cross hair where the cursor should be. Move the cursor to the absolute coordinate (0,0) and press the LMB. Now continue placing the four corners of the board geometry by clicking the LMB on the following locations: (4,0) (4,5) (0,5). Now double click on (0,0). You may want to make the board larger than you need it to be. The exact dimensions of the board can be specified later if needed. By doing so you make the layout of components easier to place in later steps.

To complete the board you will need to specify where the routing can take place and where the components can be placed on it. Go to the menu **RMB>Attributes>Create_Placement_Outline** and choose **interactive**.
You may want to change the grid so you can draw a more accurate outline. Once you are satisfied with the grid, place the cursor to the lower right hand corner of the board. Move out about .1 inches from each side of the board and press the LMB. Now place the corners of the board by clicking the LMB around the board in the appropriate places. Try to stay a uniform distance away from all sides. To complete the outline double click the LMB on on spot where the outline should be.

Next you will make the routing outline. Typically this outline should be on the outside of the placement outline. Use **RMB>Attributes>Create_Routing_Outline**. Use a similar procedure for making this outline as was done for the placement outline. Try to keep the routing outline about .05 inches off of the board outline.

If you want to add a logo to the board geometry then follow the procedure listed below, otherwise skip the next section and read on.

? [Picture of Logo](#)



Creating a Company Logo:

The designs created in this section will be made out of copper and then placed on the board geometry. This section outlines a fairly basic procedure but should provide enough information towards making a more complicated logo.

From the main menu choose **Geometries>Create_Geometry>Generic...**. A window should appear. Under 'Geometry Name:' type '**logo**' or some other appropriate name.

The current editing layer should be the layer SIGNAL_1. The current layer is displayed in the upper right hand corner of the edit window. If it is not the current layer then select **Setup>EditLayer** then select Signal_1 (Physical _1) from the list.

Setup:

You may want to change the width of the trace. To do this select **setup>linewidth** and select something higher than .01 inches.

Next you should setup the Grid to which your cursor will snap. To accomplish this select from the main menu **Setup>Grid**. Set the "X Increment' to around .01 or .05 . Leave the 'Y Increment' alone. Set the 'Display Interval' to 2. Press 'ok'.

Next you will setup the line width of the traces. Select from the Main menu **Setup>Line_Width**. Enter .01 or more depending on your needs. Try to keep the size larger than .01. Press 'ok'.

Text:

To add text to the logo you'll first need to change the height of the text. Goto **Setup>Text**... from the main menu. In this dialogue box change the following...
Height: **.062** (or different value).
Justification: **Center Center**
Press 'ok'.
Now select **RMB>Text>Add_Interactive_Text:**. Now place the mouse cursor where you want the text to start and press the LMB. Now type in the appropriate text and press enter.

If you make a mistake or you just need to move part of the design around you will first need to setup the selection filter. From the main menu choose **Setup>Select_Filter**. Once the window appears select all of the entries you can select and click on 'ok'. Now you will be able to select any of the parts displayed on the edit window. If you need to move a part just select it with the LMB. Then click the RMB and choose the appropriate command.

Drawing:

To add lines to the design use the button on the Pallet window **Addline>Vertices.**. Select points around the drawing area where appropriate and make an outline of what you want.

If you want to make a circle or polygon for your logo then choose **RMB>Shapes>Add_Polygon or Add_Circle**.

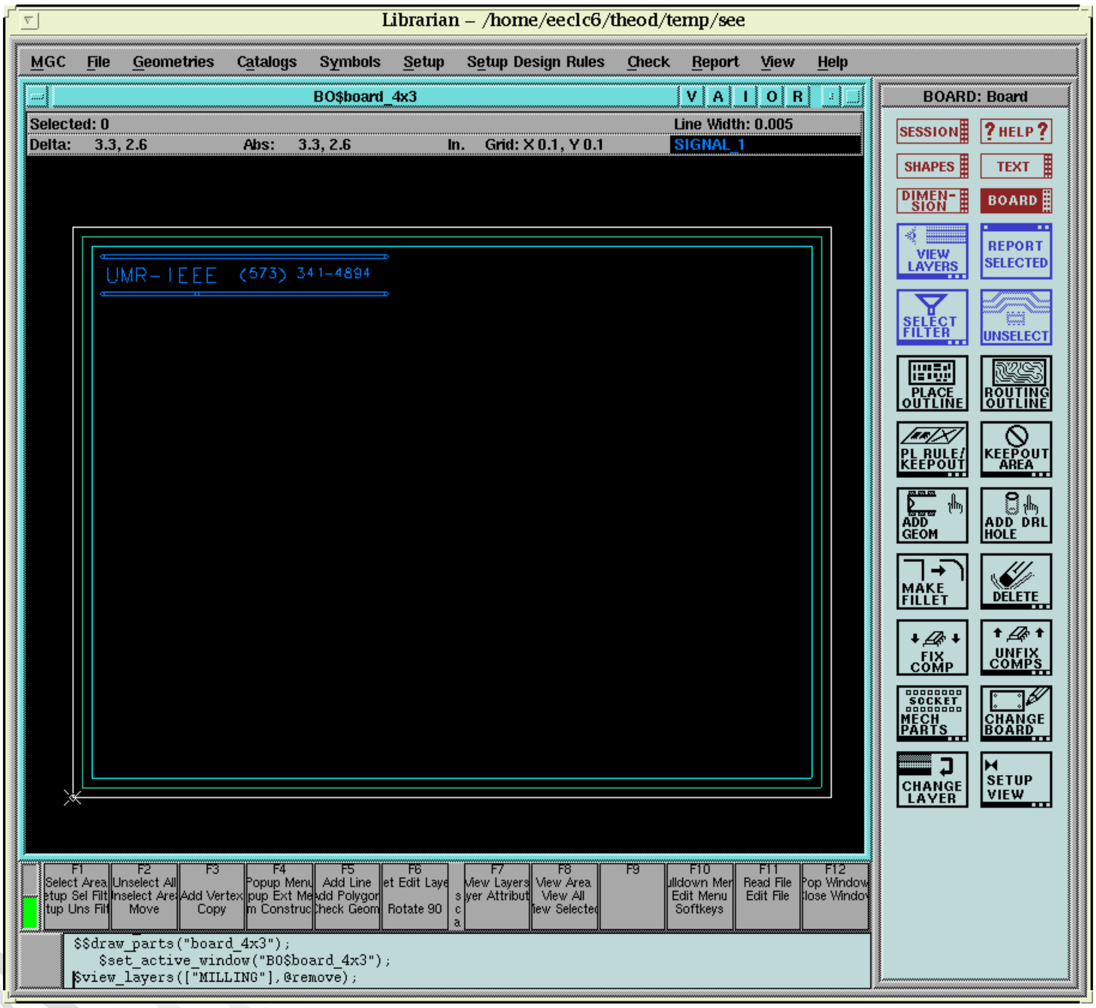
As you may have noticed there are many different ways to construct a logo. Feel free to explore the other options included in librarian. Just keep in mind that all of the pieces on the logo should be made from the SIGNAL_1 layer if you want it on the top of the board and made from SIGNAL_2 layer if it is to be on the bottom layer.

Adding Logo to Board Geometry.

To add the finished logo to the board you already constructed you will first need to switch over to the edit window with the board geometry on it. You can 'shrink' the edit window size of the windows on top of the board geometry window until you see the board OR you can go to **Geometries>Change_geometries** and select the board from the list, select 'ok' on the window that appears next.
Once you see the board geometry select

RMB>Shapes>Extended_Menu>Add_Geometry. Now fill in the box with the exact name of the logo you created. Don't change anything else in the window. Press 'ok'. Now move the cursor to where you want the logo to go. Press the LMB when you are over the right place.

? Picture of Completed Board with Logo



Making Geometries for User Defined Symbols:

Making your own symbols is a lot easier than you might think. In fact, its sometimes more convienient to make your own than it would be to search around the Board Process Libraries for the 'right' geometry.

To make a component geometry, choose **Geometries>Create_Geometries>Component** from the top menu bar. In the window that appears, enter the name of your symbol under Component Name:. The exact name of the geometry is not critical. However, you may want to include soom information about the size or dimension of the part in its name so others like it don't get confused with the one you made. For instance, if the geometry you are making is a surface mount capacitor, you may want to include the size of the pads in rests on. ie; smt_cap_20x20_mil.

For the rest of the dialog box, choose a default padstack (t032rd, sm20, t060rd,...) and then choose what type of component it is (thruhole or surface mount). You may also want to change what type of units you will be using to describe your component (Metric or English).

Once you press OK, another window will appear with the name of your component in the title bar. Zoom out to the appropriate level and change your grid spacing to something that makes sense for the type of component you're making.

Now select **RMB>Attributes>Add_Component_Body_Outline** . Now use the LMB to set the corners of the component to the desired shape. Once you have set the last corner, double click on it to complete the outline.

Now select **RMB>Attributes>Add_Placement_Region>Both_Sides**. Follow the same procedure to complete this outline.

Now you will need to place pins on the component. Do so by selecting "comp" from the palette menu. Now click on ADD_Pin. A dialog box will appear at bottom of the component window. Inside this box there will be a number under the listing "NUMERIC". You may type in what ever number you choose. The number of the geometry's pin doesn't have to match that of the component used in Design Architect. Press 'Ok' and move the cursor to the desired location. You can continue adding pins in this fashion by continuing to press 'enter' after ever pin is placed.

Once you have completed each of these steps then you can save the component by selecting File>Save_Design_All.

Catalogs and Mapping Pins:

Now that you have constructed the basic geometries of the symbols you made in Design Architect we are ready to associate the pin numbers used in Design Architect to the pins of the geometry in Librarian. This will allow traces to be connected to the parts when you enter Layout. If you make a mistake here in Librarian you may notice that some of the pins of your component don't connect properly in Layout. To avoid confusion later on, make sure you follow this procedure exactly and that you use the correct component names (case-sensitive!).

First you will need to create a catalog where all of the mapping files will be placed. The default active catalog is your 'current design' catalog and could be used for this process. However, using a separate catalog other than your design catalog will allow you to use these files in other designs and not just this one.

To create a catalog select: **Catalogs>Create_Catalog:**. A tool bar should appear at the bottom of the screen. Fill the window out as follows:

Name: **(Your_Catalog_Name)**

In Directory: **USER** (select by pressing up/down arrows)

Now we will make sure that the active catalog is the catalog you have just created. (As you build more and more designs on your account this may provide useful.) Select **Catalogs>List_Active_Catalog** . Now click on **Catalog Hierarchy**. Now click on **USER**. Now verify that the **active==>** icon is beside the catalog you want. If it is not then click the LMB on the line which has the name of your catalog. Once you have selected the proper catalog press Close.

Now we are ready to create part numbers for the physical geometries we have just created. These files will allow traces to be routed to the parts used in your design. If you do not need traces routed to a particular part then you can skip this step without ruining your design.

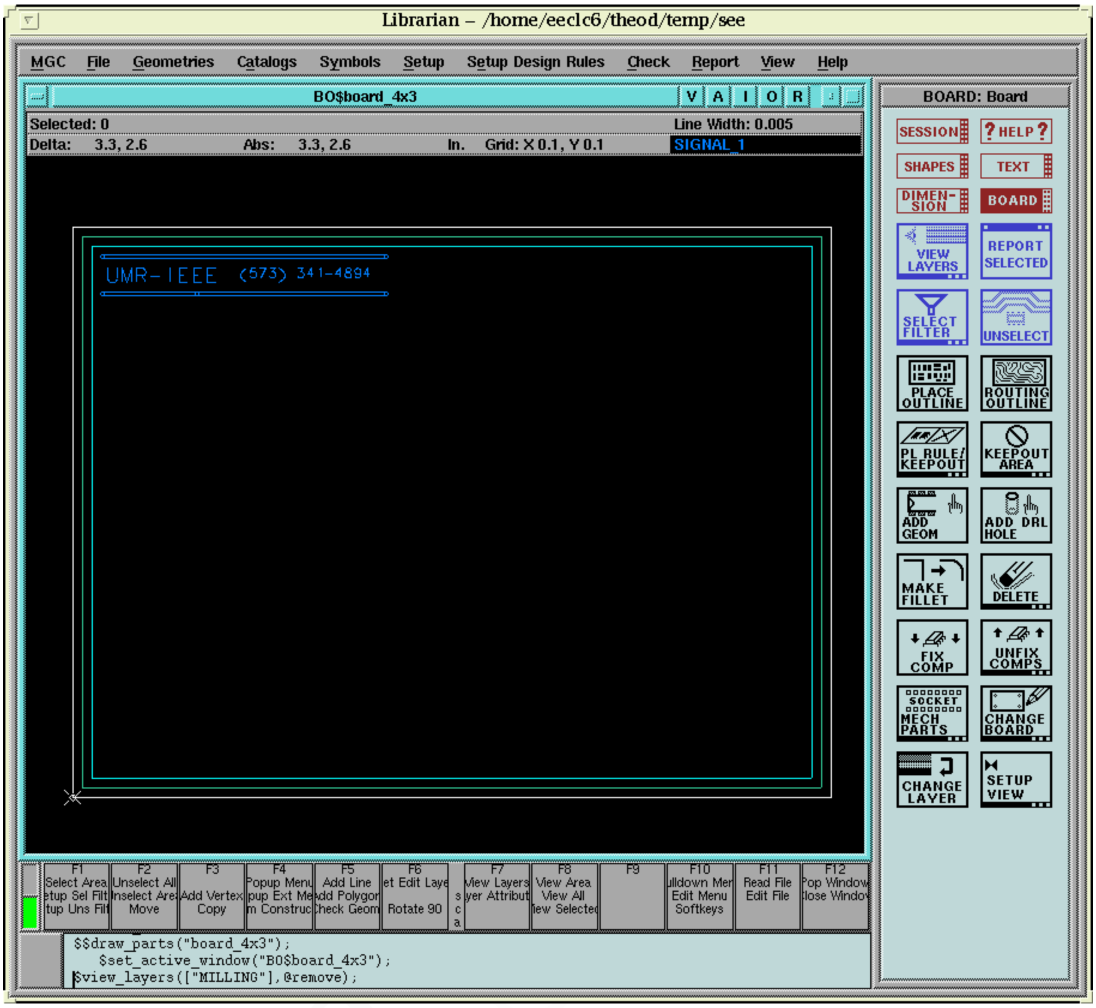
To create a part number select **Catalogs>Create_Part_Number**. If you are following the IEEE-SEE project then fill out the window as follows:

Part Number: **PN-6pin_res**
Catalog Name: **(should already be there)**
Geometry: **6pin_res** (make sure this is the exact name)
Mapping: **(leave blank)**

Replace Part: **Yes**

Two windows should now be displayed on the screen. The one on the left is the mapping window and the one of the right is the geometry window.

? Picture of Completed Mapping File of 6pin_res



In the left window select **RMB>Map_Logic_Symbol...**. Now fill out the window as follows:

Comp Property

Property Value: **6pin_res** (use exact name)

Map Symbol Count: **1**

Map Symbol Name: **6pin_res** (name not critical)

Now we are ready to add logic pins. You could also add power pins if you wanted to, however, this is typically used for designs that have dedicated power and ground planes. To add logic pins select in the left window **RMB>Add_Logic_Pins**. Now fill out the window as follows:

Create Pin Name: 1

Swap Code: 1

Create Pin Name: 2
Swap Code: 1

Create Pin Name: 3
Swap Code: 3

Create Pin Name: 4
Swap Code: 3

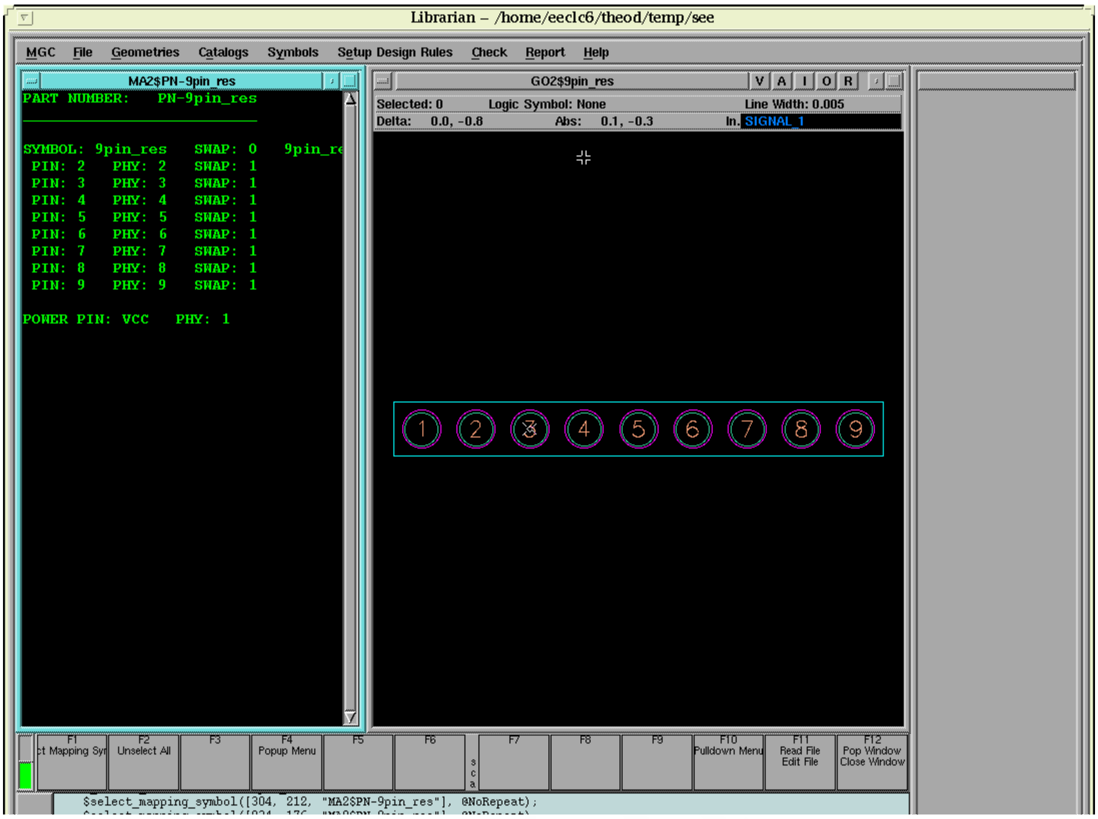
Create Pin Name: 5
Swap Code: 5

Create Pin Name: 6
Swap Code: 5

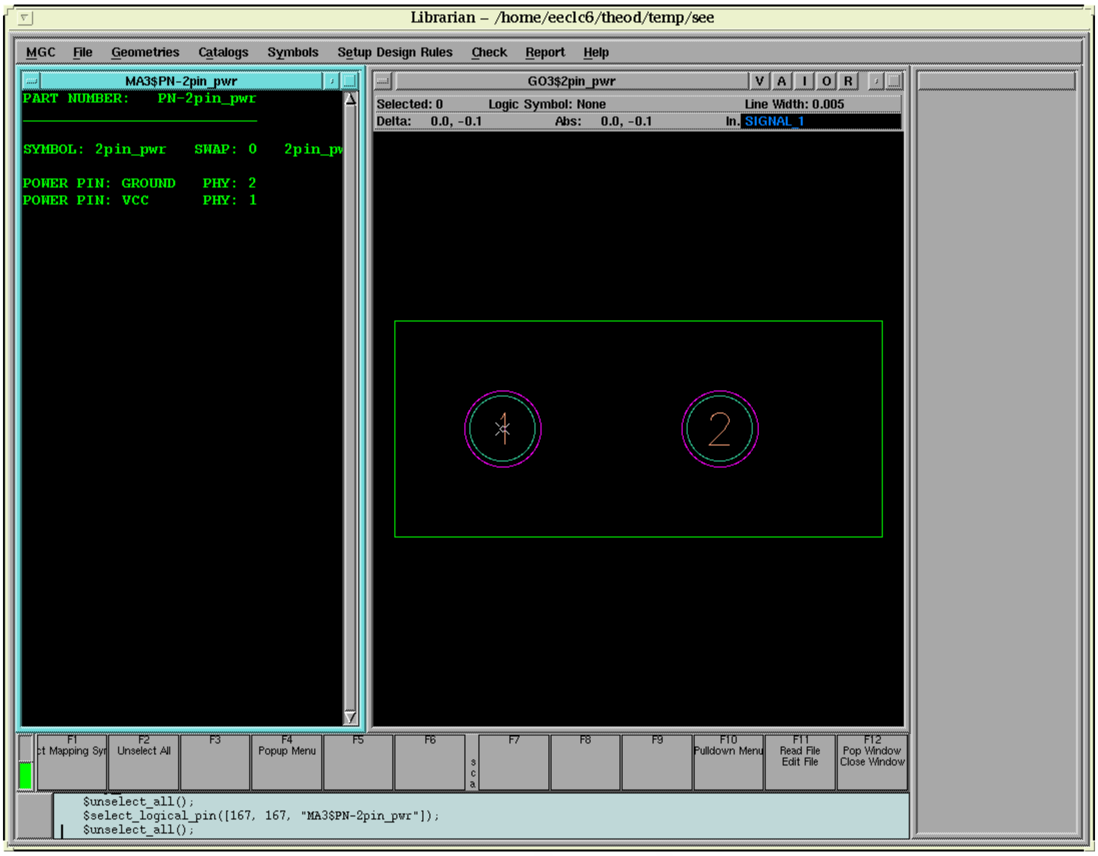
After you have completed this window you should see a list of all the logic pins you have just made. The swap codes to the right of the pin name indicates which pins can be swapped by the auto-router used in Layout to save unnessary routing. Two pins with the same swap code can be switched with each other. The swap number is completely arbitrary except that they are not zero. A swap code of zero means that the pin shouldn't be switched with anything.
Now we are ready to associate the pins in the left window with the pins in the geometry window. Select from the left window **RMB>Map_Pins:**. Now click on a pin in the left window and then click on its corresponding pin in the right window. Continue clicking until all pins have been matched up.

Now check your part number by selecting **Check>Part_Number>Active_Part_Number**. Get rid of the report window and fix any errors if present. Once your part number is correct, save the design by selecting **File>Save>Design_All**.

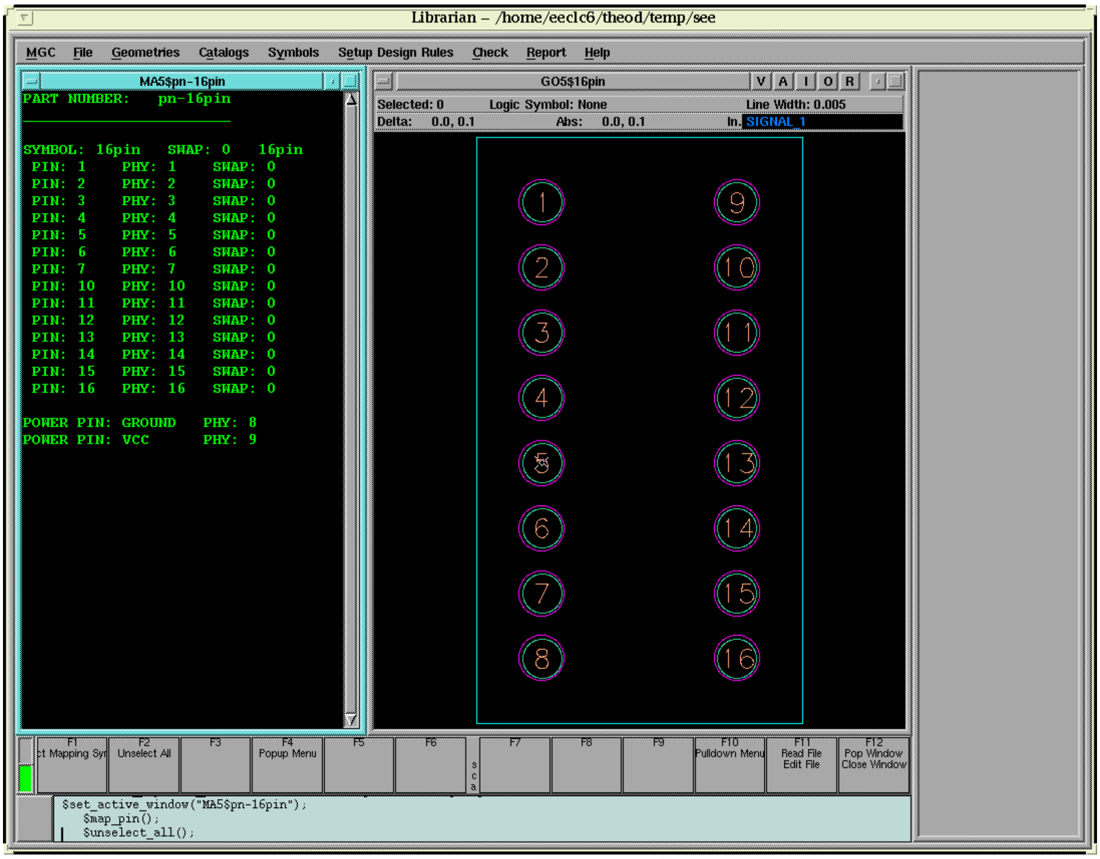
For the IEEE-SEE project the following part numbers and mapping files were created.
? Mapping File of 9pin_res



? [Mapping File of 2pin_pwr]()



? Mapping File of 16pin



Saving Geometries and Saving Design:

Now you will need to create a backup of your work. Do so by going to **File>Save>ASCII_Geometries...**

Fill the dialogue box out like this...
Geometries to save: **All Geometries (or Specific Geometries)**
Separate File for Each
Library to Store the Geometry: **Design**
Replace Existing File(s)

To save the work on the design go to **File>Save>Design**

After a few messages appear and confirm the save, you can leave librarian. You are now ready to use Package.

? Next Part of Tutorial: Package
? Back to Main Page: Go to Beginning

Board Station Tutorial

Part 5/7

Package

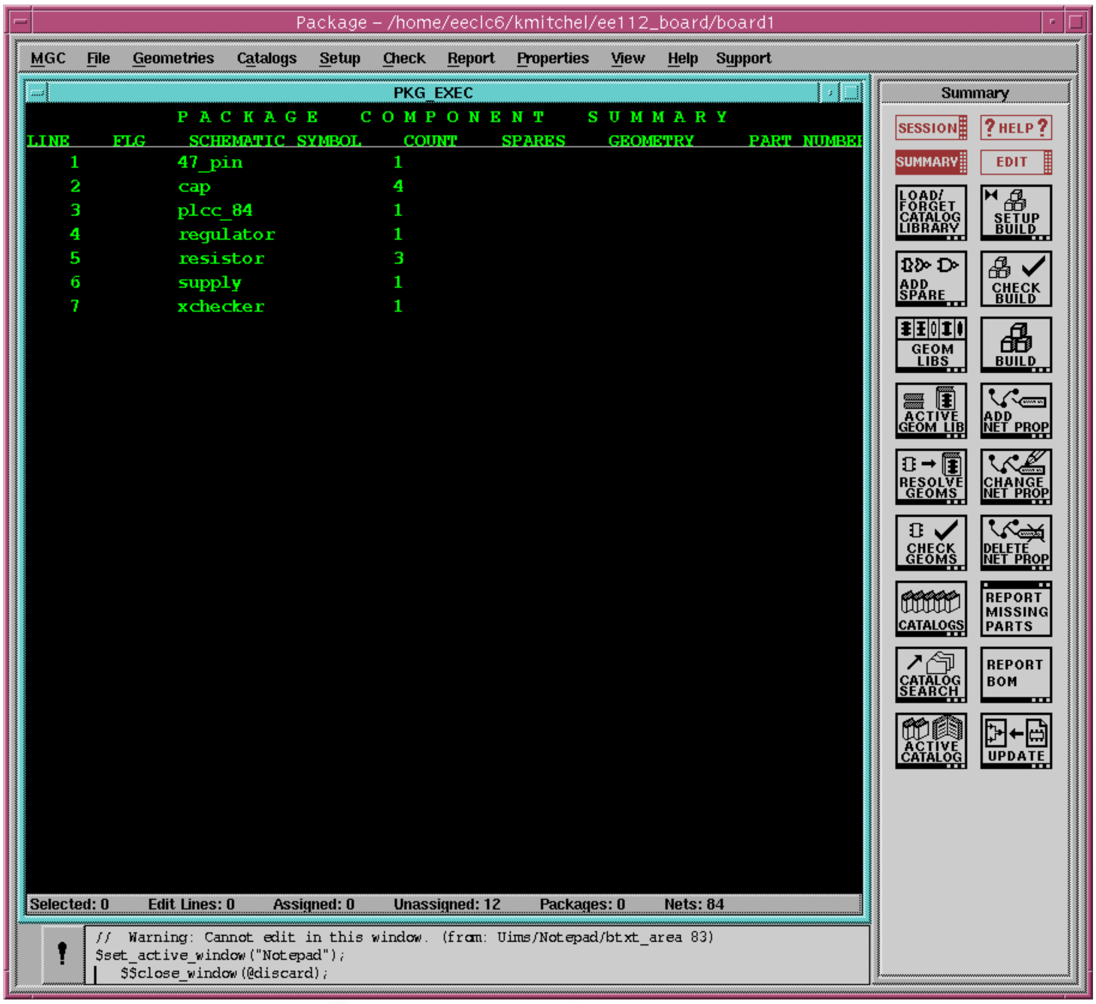
The purpose of this section of the tutorial is to run Package, a program which collects all of the information from Design Architect and Librarian and organizes them into a format which can be easily read by the next two programs of the tutorial, Layout and Fablink. First you will read in all of the geometries you have made, then Package will organize all of the information.

To run Package first make sure that the working directory is the one which contains the directory of your design. If you are following this tutorial exactly, then that would be the PCB directory. Type 'sul' and 'swd' once inside this directory.

Execute the program on your design by typing: **package Design1** . The program will load and a series of warning messages that you should ignore. It may take a few minutes to load.

After the messages are gone get rid of any report windows. The screen should look something like the link below.

? Picture of Package



To the far right of the screen there should be a button labeled '**Active Catalogs**'. Click on this button. You may need to expand the size of the Package window. Just go to the extreme lower left corner of the window and use the LMB to expand the window size.

You should see a large beige dialogue window on the screen. At the bottom of the screen you should see a small box next to the words 'Catalog Hierarchy'. Click on this box.
Now you should see a list of all of the catalogs currently on the system. Go to '**Design Catalogs**' and click with the LMB.
Now you should see the **ACTIVE => design Status: UNREAD**. Click on 'READ', which is in the lower left hand side of the window. After you are done, 'Close' this window.

Now that the geometries you created are loaded onto the system you will need to combine them with the ones already there. Go to the right side of the screen and click on the button named '**Update**'. Use the default values that appear in the window that comes up. After the program is complete you should see more information listed on the screen than you had before. This information includes the part numbers, mapping files and the geometries of the parts used.

Now you are ready for the next step of the design process. Go to the top menu and choose **Save>Design_All**. Save it with Back Annotate and view point. Once completed, exit Package.

? Next Part of Tutorial: Layout
? Back to Main Page: Go to Beginning

Board Station Tutorial

Part 6/7

Layout

The purpose of this program is to construct the PCB into its final form.

As was done before, make sure that the working directory is set to the one that has the directory of your design in it. Use 'sul' and 'swd' once in this directory.

Execute the program on your design by typing: **layout Design1** . Once the program is loaded clear the report window. Ignore any 'warnings' you encounter in the report window, they will be resolved later. If you get any 'errors' then you have done something wrong and will need to backtrack to previous steps.

A picture of the board should appear on the screen. In order to view everything on the board select **View>Layers** from the top menu. Now select **All_visible** in the window that appears. Close the window.

Placing Components:

Now you will place all of the components on the board. Select **RMB>Auto_Placement>Place_Components**. Keep all of the default values except make sure that the will be placed on the '**top only**'. Press 'ok' once you are done and like the settings in the window. The program will now place all of the parts according to what it feels is the smallest configuration of components. When the program is done you should see all of the components used in your design on the board. You may have noticed that this program does a poor job of placing components. To move the parts where they should be go to **Setup>Select_Filter** and choose components only. Now click on a component and use **RMB>Move or RMB>Rotate** to get the parts to a desirable location. If you violate any placement rules(putting one part atop another) then it will return the part its last location.
Once you are satisfied with the location of the parts you are then ready to set down some design rules and begin routing.

Creating Routing, Layer, Net and Grid Rules.

To create routing rules select **Setup_Routing>Routing_Rules**. Fill out the dialogue box as follows:

Trace Grid X: **.005** Y: **.005**
Pin Grid X: **0** Y: **0**
Via Grid X: **0** Y: **0**

T-Junctions Allowed

Diagonals Allowed
Trace Vertex Bends: **Orthogonal and Diagonal**
Pad and Via Entry: **Orthogonal and Diagonal**

Route Connections By: **Length**

Setting Up Net Rules:

Before you take this design to the next level a few design parameters will need to be put in place. These rules will govern how Layout and Fablink will place components and rout traces.
To setup these rules choose from the main menu **Setup_Routing>Change>Net_Rules...**.
Now choose **Create_New_Net_Type** and then make **DEFAULT_NET_TYPE** the name.
Now choose **Setup rules...**. Now fill in the categories like this...

Trace Width **.012**
Pin to Pin Clearance: **.013**
Via to Via Clearance: **.013**

Pin to Trace Clearance: **.013**
Via to Trace Clearance: **.013**

Pin to Via Clearance: **.013**
Trace to Trace Clearance: **.013**

For Interactive Routing Vias: **v050rd**
For Interactive Routing Vias for Auto Routing: **v050rd**

Available Routing Layers: **Physical_1 and Physical_2**
(Choose both by holding down CTRL button and click once on each selection)

If you need to alter the grid to which the router will adhere to then select **RMB>Auto_Routing>Change_Routing_Grid>Change_Routing_Grid...**. Fill out the window to your specifications. If you change your mind then select **RMB>Auto_Routing>Change_Routing_Grid>Reset_to_Design_Rules...**.

To change the Layer rules select the following: **Setup_Routing>Layer_Rules...**. Make sure that Physical_1 (top layer) is disabled for the meantime and that the preferred direction of Physical_2 (the bottom layer) is set to 'ALL'. This done so that the majority of the traces will be on the side that is most easily soldered.
After all of the traces that can be made have been, the physical_1 layer should be reactivated. This will cut down on the number of vias used to make the circuit board ($$$).

Routing Traces:

Once you are confident that the rules meet your requirements and that the parts are in their best location then you are ready to begin the Auto-Routing process.
Begin by selecting **RMB>Auto_Routing>Start_Auto router**. Fill out the window as listed below.

Breakout
Number of Passes: **1**
Automatic
Number of Passes: **5**
Pattern
Number of Passes: **5**
Manufacturing
Number of Passes: **5**
The rest of the window should be left at the default values. Click on 'ok' once you are done.

Once this program is run you should see displayed a scrolling report displayed across the screen. At the end of the report note the number of unroutes. A running total is displayed above the scrolling window (you may find it easier to read). If there are some unroutes then you may need to run the Auto-Router again. If this is the case then you may want to delete all of the traces it has made by first changing the selection filter at **Setup>Select_Filter** by selecting only traces. You don't necessarily have to delete all of them either. Then you need to select all traces on the board and press delete. Now you can either use the same defaults for the traces as you used above or you may want to reduce the trace width and use no grid for traces. Repeat as necessary.

When all of the traces have been routed close the report window and verify that the traces have been made. If the are there then you are ready for the final stage in the design process, Fablink.

? Next Part of Tutorial: Fablink
? Back to Main Page: Go to Beginning

Board Station Tutorial

Part 8/8

Fablink

The purpose of this program is to make information files that can be used with the routing machine in ERL. To accomplish this task several files will be made. (One for all of the drill holes, one for the milling data, and one for the every trace layer used. There will also be a few report files made for you reference.) All of these files will be automatically placed in the /pcb/MFG directory of your design.

To enter Fablink type this at the command prompt: **fablink Design1**
After Fablink has loaded close any report windows and ignore any warnings about test points.

Drilling Data

To create the drilling data first select **RMB>Drill>Change_Drill_Table>Fill_Drill_Table**. Don't change any of the values in the window and press 'ok'.

Now we will create the drill data and verify that it looks correct. Select **RMB>Drill>Create_Drill_Data** . Now fill out the window with the values given below.

Drill File Format: **Excellon**
Generate for: **Board**
Drill Character Set: **ASCII**
Mirror: **OFF**
Drill Hole Types: **Both**
Output Hole Types: **All Types**

A report window should now be displayed, close it to continue.

To verify that the drill data worked properly choose: **RMB>Drill>Simulate>Drill_Data...** . You should see the drill holes and the path that the drill will take outlined in red. Close the window when you feel that this looks fine. If this doesn't look fine to you then maybe you made an error in librarian when you defined some components or maybe the drill table was filled out incorrectly.

Now go to **Report>Drill_Table** and **include drill format** and **save and display report**.

Artwork Data (Traces and Logos)

Now we will create artwork data that will represent every copper trace and every other shape made from copper. There will be one file created for every trace layer in your design.

Begin by selecting **RMB>Artwork>Change_Apeture_Table>Fill_Apeture_table**

Next go to **RMB>Artwork>Change_Artwork_Format** and set **Plot_Offsets** to **Manual** with **(0,0).**.

Then choose **RMB>Artwork>Create_Artwork_Data**

Fill out the dialogue window as follows:

File Format: **Gerber**
Character Set: **ASCII**

Artwork Number: **ALL**
Output All Pins, Vias

Resize, Rescale Artwork: **NO**

After filling this out go to **RMB>Artwork>Simulate_Artwork_Data**
Choose the first one (Signal_1 or Physical_1). After this is done another window will appear on the screen. This window will display the top layer's traces and logos made from copper. Close the Artwork window once you are done looking at it.
Now follow the same procedure but for the other layer(Signal_2 Physical_1).

To let the milling machine operator know what type of tools to use, you will need to create a file that contains trace information. Go to **Report>Aperture_Table** and select...
Include Artwork Format
Save and Display Report

Milling Data

Now we will make the milling data needed for cutting the board outline. Make sure the small box in the upper right hand corner of the main edit window says 'milling' and not some other layer name. If it doesn't say 'milling' then change it by selecting >Setup>Change_Edit_layer> and then click on the name milling which is in the left column of names. Verify that the name 'milling' appears in the small window. If it doesn't the the program will not recognize the milling outline until you fix it.

Begin by selecting **RMB>Milling>Change_Tool**. Select..

Tool Diameter: **.040 (or .080)**
Tool compensation: **Right**
Plunge at : Absolute Location
Digitize Point at: Absolute Location

If you want to follow the outline of the board exactly then choose ***Nearest Vertex*** *for 'Plunge at' and 'Digitize Point'. For the IEEE-SEE project the final size of the board was unknown and therefore an exact board geometry was not made in Librarian.*

After closing this window select **RMB>Milling>Start_Tool_Path**. Now select a corner of the board where you want to start cutting and click the LMB. Now click on the corners of the board where you want it to be cut at. Be sure to go in a counter-clockwise direction. Once you have made it around the board cancel the tool bar or use the shortcut for escape.

Now select **RMB>Milling>Change_Milling_Table>Fill_Milling_Table**

Now select **RMB>Milling>Create_Milling_Data**. Keep the defaults in the window and press 'ok'.

Now go to **Report>Milling_Table** and **include milling format** and **Save & Display Report**. Close the report window.

You are now done with this portion of the design. Save and Back Annotate the design as you have done before. Close the Fablink window. All of your data files are now in the /pcb/mfg directory of your design. Next you will take the data files to the milling machine for board fabrication. See the milling operator for details.

? Back to Main Page: Go to Beginning

射 频 和 天 线 设 计 培 训 课 程 推 荐

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立，致力并专注于微波、射频、天线设计研发人才的培养；我们于 2006 年整合合并微波 EDA 网(www.mweda.com)，现已发展成为国内最大的微波射频和天线设计人才培养基地，成功推出多套微波射频以及天线设计经典培训课程和 ADS、HFSS 等专业软件使用培训课程，广受客户好评；并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书，帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司，以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

易迪拓培训课程列表：http://www.edatop.com/peixun/rfe/129.html

射频工程师养成培训课程套装

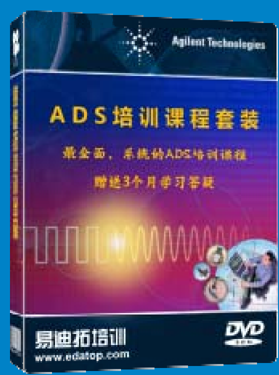
该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材；旨在引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和研发设计能力。通过套装的学习，能够让学员完全达到和胜任一个合格的射频工程师的要求…

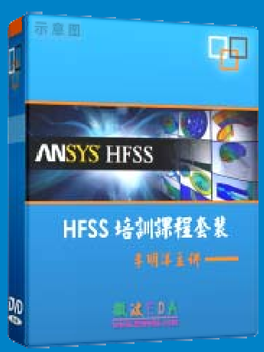
课程网址：http://www.edatop.com/peixun/rfe/110.html

ADS 学习培训课程套装

该套装是迄今国内最全面、最权威的 ADS 培训教程，共包含 10 门 ADS 学习培训课程。课程是由具有多年 ADS 使用经验的微波射频与通信系统设计领域资深专家讲解，并多结合设计实例，由浅入深、详细而又全面地讲解了 ADS 在微波射频电路设计、通信系统设计和电磁仿真设计方面的内容。能让您在最短的时间内学会使用 ADS，迅速提升个人技术能力，把 ADS 真正应用到实际研发工作中去，成为 ADS 设计专家...

课程网址： http://www.edatop.com/peixun/ads/13.html





HFSS 学习培训课程套装

该套课程套装包含了本站全部 HFSS 培训课程，是迄今国内最全面、最专业的 HFSS 培训教程套装，可以帮助您从零开始，全面深入学习 HFSS 的各项功能和在多个方面的工程应用。购买套装，更可超值赠送 3 个月免费学习答疑，随时解答您学习过程中遇到的棘手问题，让您的 HFSS 学习更加轻松顺畅…

课程网址：http://www.edatop.com/peixun/hfss/11.html

CST 学习培训课程套装

该培训套装由易迪拓培训联合微波 EDA 网共同推出，是最全面、系统、专业的 CST 微波工作室培训课程套装，所有课程都由经验丰富的专家授课，视频教学，可以帮助您从零开始，全面系统地学习 CST 微波工作的各项功能及其在微波射频、天线设计等领域的设计应用。且购买该套装，还可超值赠送 3 个月免费学习答疑…

课程网址：http://www.edatop.com/peixun/cst/24.html

HFSS 天线设计培训课程套装

套装包含 6 门视频课程和 1 本图书，课程从基础讲起，内容由浅入深，理论介绍和实际操作讲解相结合，全面系统的讲解了 HFSS 天线设计的全过程。是国内最全面、最专业的 HFSS 天线设计课程，可以帮助您快速学习掌握如何使用 HFSS 设计天线，让天线设计不再难…

课程网址：http://www.edatop.com/peixun/hfss/122.html

13.56MHz NFC/RFID 线圈天线设计培训课程套装

套装包含 4 门视频培训课程，培训将 13.56MHz 线圈天线设计原理和仿真设计实践相结合，全面系统地讲解了 13.56MHz 线圈天线的工作原理、设计方法、设计考量以及使用 HFSS 和 CST 仿真分析线圈天线的具体操作，同时还介绍了 13.56MHz 线圈天线匹配电路的设计和调试。通过该套课程的学习，可以帮助您快速学习掌握 13.56MHz 线圈天线及其匹配电路的原理、设计和调试…

详情浏览：http://www.edatop.com/peixun/antenna/116.html

我们的课程优势：

※ 成立于 2004 年，10 多年丰富的行业经验，

※ 一直致力并专注于微波射频和天线设计工程师的培养，更了解该行业对人才的要求

※ 经验丰富的一线资深工程师讲授，结合实际工程案例，直观、实用、易学

联系我们：

※ 易迪拓培训官网：http://www.edatop.com

※ 微波 EDA 网：http://www.mweda.com

※ 官方淘宝店：http://shop36920890.taobao.com