



Guide to Mentor Graphics *PADS* 2005  
Department of Electrical and Computer Engineering  
Fall 2006

(last revised 11/2/06)

## Summary

This is a tutorial on how to design, layout, and generate fabrication files for a Printed Circuit Board (PCB) using Mentor Graphics PADS 2005 SPac2. PADS Logic is a schematic entry tool where symbols and net connections are entered. PADS Layout is a physical design tool where the board outline, component placement, traces, and power/ground planes are entered to physically implement the design. PADS Layout also creates Computer Aided Manufacturing (CAM) files that are used by the PCB fabrication vendor to ultimately build the printed circuit board.

This tutorial will walk through an example PCB design from start to finish in order to explain all of the steps in the design. The board will be a simple 4-layer board containing a 52-pin, surface mount (SMT) connector in addition to a variety of SMT resistors.

It should be noted that PADS is a *design entry* tool, not a *design* tool. The electrical design of the circuitry is designed elsewhere (pen and paper, SPICE, etc...). PADS is only used once the design is finalized and needs to be physically implemented. For this tutorial, it is assumed that the design has been completed and is ready to be implemented using a PCB.

The following lists the steps that will be covered in this tutorial:

- Step 1: Part Setup
- Step 2 : Schematic entry
- Step 3 : Layout
- Step 4 : Design Rule Check (DRC)
- Step 5 : Generating Fabrication Files

## Step 1 : Part Setup

This section describes setting up the symbols, footprints, and electrical data for the parts that will be used in the design. These parts will go into a custom part Library that can be used on subsequent designs.

A “Part” refers to an object that can be placed in the schematic and layout. The most common *part* is an electrical component such as a resistor or connector. In order for the part to be used in the design, it must be setup in the Part Library. There are three data items that come together to form a *part*.

A Part Contains:

- 1) CAE Decal - This is the schematic symbol that is used in PADS Logic.
- 2) PCB Decal - This is the layout footprint (or geometry) that is used in PADS Layout.
- 3) Electrical Data - This contains the specific information for the unique part. The electrical data links how the pins on the CAE Decal map to the pads on the PCB Decal. It also contains the value (i.e., 50 ohms) of the part, the part number, manufacturer, and any other information that is unique to this part.

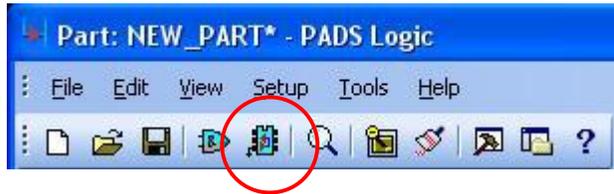
NOTE: A CAE Decal and PCB Decal can be used in multiple parts. For example, there may be thousands of resistors that use the same CAE Decal and same PCB Decal. However, each resistor value would be setup as a unique Part that contains a unique value, manufacturer, etc... PADS contains some of the more commonly used CAE Decal and PCB Decals such as resistors, SOIC’s, etc... For parts that are not in the PADS common database, the engineer must manually generate the CAE and PCB Decal.

### **Exercise 1(a) - Create a Part for a 20k Ohm, 0402, SMT Resistor**

For this part, the symbol and footprint already exist in the PADS database. We will create a new unique part and associate the existing symbol and footprint to the new part. In addition, we will add information on the value and manufacturer.

- Start PADS Logic
  - Start → Programs → Mentor Graphics SDD → PADS2005 SP2 → System Design → PADS Logic
  - At the “Welcome to PADS Logic” screen, click “Start a new design”
- Start the Part Editor
  - On the “Tools” pull down menu, click “Part Editor”

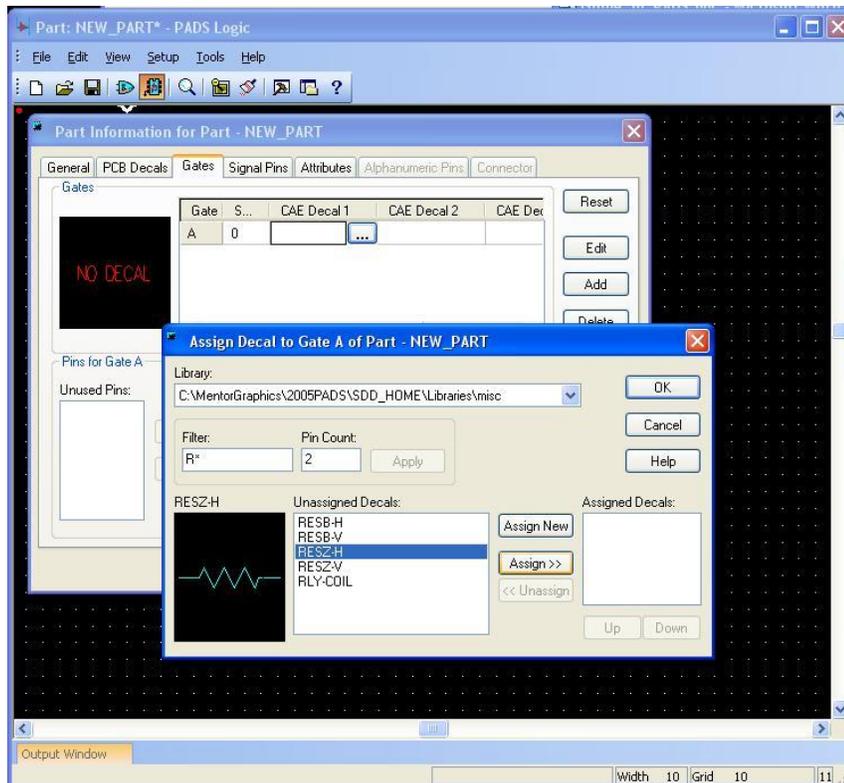
- Click on the “Edit Electrical” icon in the Part Editor Window



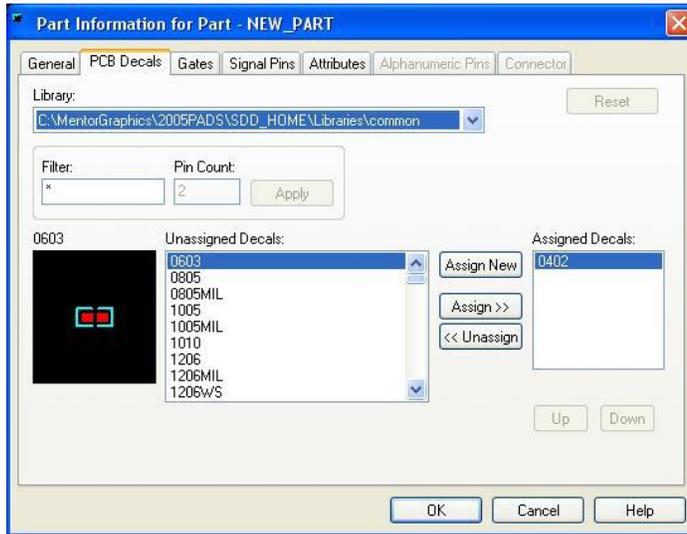
This brings up the “Part Information” dialog.

- Click on the “Gates” tab to add a symbol for this part:

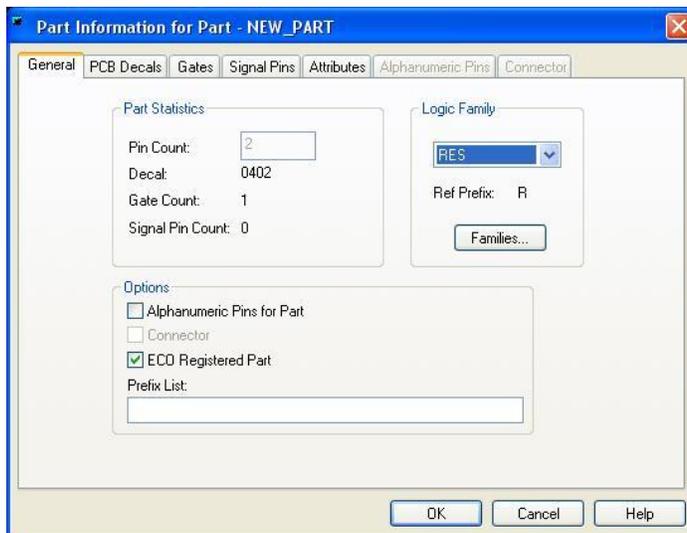
- Click the “Add” button
- Double Click in the “CAE Decal 1” field
- Click the “...” button to browse for a CAE Decal in the PADS database
- In the “Assign Decal to Gate” window, choose the “misc” library from the Library pull down
- To search for a resistor symbol, enter “R\*” in the Filter field and “2” in the Pin Count field, click apply
- Highlight the “RESZ-H” Decal and click “Assign >>”
- Click “OK” to close the “Assign Decal to Gate” dialog



- Click on the “PCB Decals” tab to add a footprint for this part:
  - Choose the “common” library from the Library pull down
  - To search for an 0402 footprint, enter “\*” in the Filter field and “2” in the Pin Count field, click apply.
  - Highlight the “0402” Decal and click “Assign >>”



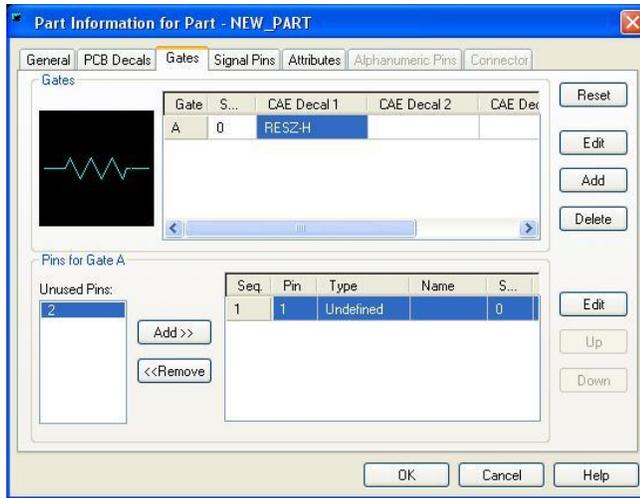
- Click on the “General” tab to assign the Logic Family for this part:
  - On the “Logic Family” pull down, choose “RES”. This will give the part a reference designator of “Rxxx”
  - Check the “ECO Registered Part” box (if not already checked).



- Click on the “Gates” tab again to map the Symbol pins to the Footprint pins

- Highlight “1” in the “Unused Pins” field and click “Add >>”
- Highlight “2” in the “Unused Pins” field and click “Add >>”

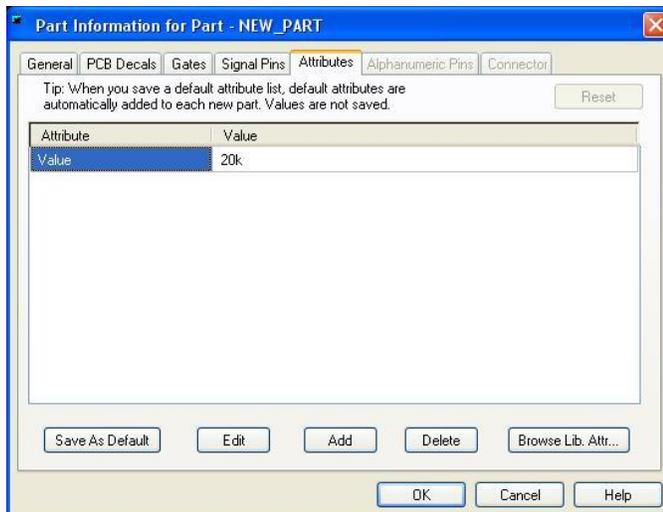
We can leave the Name Field blank since this is a very simple, bidirectional, component. If you wish, you can enter a descriptive Name such as “Input 1”.



- Click on the “Attributes” tab to assign additional information for this part.

- Click the “Add >>” button, this puts the cursor in the “Attribute” field
- Click the “Browse Lib. Attr” button
- Select “Value” from the Attribute list and Click “OK”
- Double Click in the “Value” field, type “20k”, then hit TAB to end the entry mode

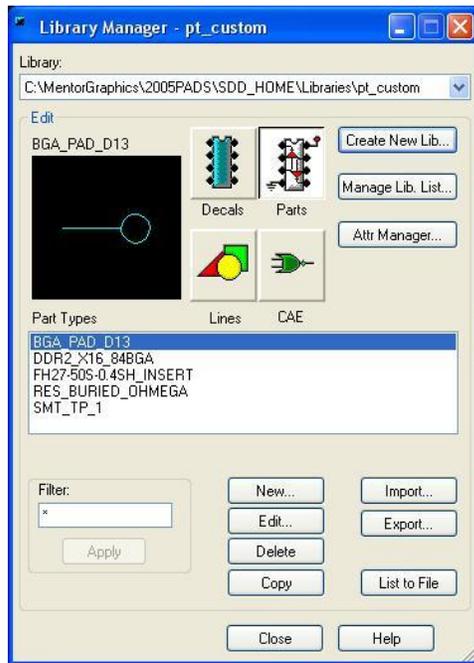
At this point you can add additional information such as Manufacturer, Part #, Cost, etc... as specific attributes for this part.



- Click OK on the “Part Information” dialog

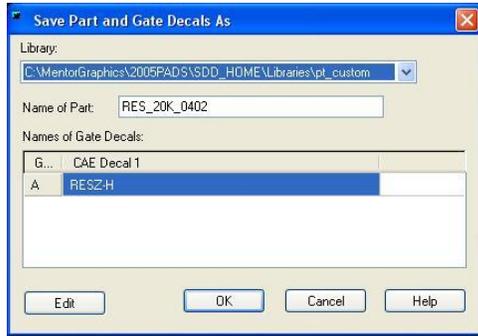
We now want to save the Part. We will create a new Library to hold all of the parts we are setting up. This library can be used on subsequent PCB designs.

- On the “File” pull down menu, click “Library” to bring up the Library Manager window.
- Click the “Create New Lib...” button. Browse to where you want the library to be located and give it a descriptive name. For this project, we will create a new library called “pt\_custom”. Click “OK” to accept the new library creation.



Now that we have a new custom Library to hold all of our parts, we can save the 20k ohm resistor we’ve just created.

- On the “File” pull down, click “Save”.
- Select the custom library we’ve just created (ex. pt\_custom) from the Library pull down.
- Give the part a unique name such as “RES\_20k\_0402”.
- Click the “OK” button.



- On the “File” pull down, click “Exit Part Editor” to return to PADS Logic.

We have now setup a new custom library (ex. pt\_custom) which contains our 20k ohm, 0402 resistor. This resistor can now be added to a schematic and placed in a layout. If you want to verify that the part exists, on the “File” pull down menu in PADS Logic, click on “Library”. Click the “Parts” button. The “RES\_20k\_0402” part will show up in the “Part Types” list in your custom library.

We will add more parts to the custom library we just created. Make sure to keep this library in a secure location. You will be using continually using this library.

### **Exercise 1(b) - Create a Part for a 75 Ohm, 0402, SMT Resistor**

Repeat all of the steps from exercise 1(a) to create another unique part for a 75ohm, 0402, SMT resistor. The only difference will be that the “Value” attribute will be 75ohms, and the part name will be “RES\_75\_0402”.

After creation, verify that both the “RES\_20k\_0402” and “RES\_75\_0402” parts show up in the Library Manager in your custom library.

### **Exercise 1(c) - Create a Part for a 52-pin, 0.4mm pitch, SMT, ZIF Connector**

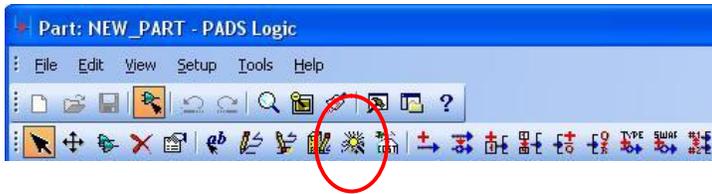
Now we will create a part for a connector that doesn’t have a symbol or footprint that can be used from the PADS database. This means we must create the CAE and PCB Decals from scratch. We will create the Part and CAE Decal using PADS Logic. We will then create the PCB Decal and complete the pin mappings using PCB Layout.

The information to create the PCB footprint comes from the manufacturer datasheet. We will be creating a Part for the *Hirose, 0.4mm, 50-pin, FC ZIF Connector*. The pin has two mechanical pins also so the total pin count is 52 pins.

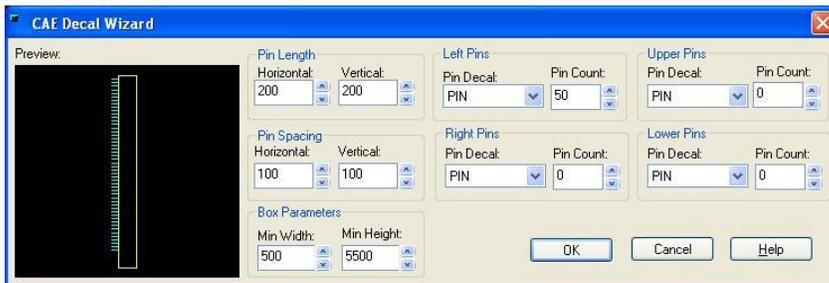
- In PADS Logic, on the *Tools* pull down, select *Part Editor*
- Click the “Edit Gate Decal” button



- You will receive a warning that a new decal is being created, click “OK”.
- Click the “CAE Decal Wizard” button.



- Click the “CAE Decal Wizard” button.



- Enter “50” for the “Left Pin” Pin Count, enter “2” for the “Lower”, and “0” for the “Upper” and “Right” Pin Counts.
- Enter “500” for the “Min Width” and “5500” for the “Min Height” in Box Parameters.
- Click “OK”

- Assign Pin Names to the gate.

- Click the “Set Pin Name” button.



- Enter “PIN\_1” and click “OK” (you can give more descriptive names in you’d like)

Now you are in “Set Pin Name” mode, when you click on a pin, it will assign the name to that pin and enter it in the symbol body. When you click on the next pin, it will auto increment the pin name to PIN\_2.

- Click on the uppermost pin of the symbol, “PIN\_1” will appear.
- Click on the 2<sup>nd</sup> to the uppermost pin of the symbol, “PIN\_2” will appear
- Continue to assign pin names on the rest of the pins on the symbol, when complete, Right Click and select “cancel”.
- Click the “Change Pin Name” button and change the names of the bottom two pins from “PIN\_51” and “PIN\_52” to “MECH\_0” and “MECH\_1”. Right click and select “cancel” when complete.

- Assign Pin Numbers to the gate.

- Click the “Set Pin Number” button.



- Enter “1” and click “OK”.

Now you are in “Set Pin Number” mode, when you click on a pin, it will assign the number to that pin and enter it above the pin. When you click on the next pin, it will auto increment the pin number to 2.

- Click on the uppermost pin of the symbol, “1” will appear.
- Click on the 2<sup>nd</sup> to the uppermost pin of the symbol, “2” will appear.
- Continue to assign pin numbers on the rest of the pins, when complete, Right Click and select “cancel”.

If you wish to manually change the pin numbers, you can choose the “Change Pin Number” button.

- Name the CAE Decal and Part Type.

- On the “File” pull down, click “Return to Part”.
- Click “Yes” to save the changes to the CAE Decal.
- Back in the Part Editor, on the “File” pull down, click “Save As”.
- Choose your custom library in the Library pull down (ex pt\_custom)
- In the “Name of Part:” field, enter a name for this part. Give the name “FH27-50S-0.4SH\_CONN”.
- Double Click in the “CAE Decal” field to give a name to the CAE Decal you’ve just created. Give the name “FH27-50S-0.4SH\_CONN\_SYMBOL”.
- You will get an error window that says no PCB Decal is assigned. Close this window.

- Assign Electrical properties to the part

- Click on the “Edit Electrical” button
- On the “General” tab, go to the “Logic Family” pull down and select “CON”. This will give a Ref Prefix of J. Make sure that the ‘ECO Registered Part’ box is checked.
- On the “Gates” tab, verify that the newly created Decal has been assigned and that the Signal Names/Numbers are correct.
- On the “Attributes” tab, click the “Add >>” button, this puts the cursor in the “Attribute” field.
- Click the “Browse Lib. Attr” button.
- Select “Part Number” from the Attribute list and Click “OK”. Enter “FH27-50S-0.4SH” for the Value.
- Click the “Browse Lib. Attr” button again.
- Select “Manufacturer #1” from the Attribute list and Click “OK”. Enter “Hirose” for the Value.
- Click “OK” on the “Part Information” window to return to the Part Editor.

- Click the “Save” button in the Part Editor to save the part

- On the “File” pull down, click “Exit Part Editor”

At this point, the Part, CAE Decal, and some of the Electrical Properties have been created. We need to complete the Part by creating the PCB Decal in PADS Layout.

- Start PADS Layout

- Start → Programs → Mentor Graphics SDD → PADS2005 SP2 → PCB Layout → PADS Layout
- At the “Welcome to PADS Layout” screen, click “Start a new design”

- Start the PCB Decal Editor

- On the “Tools” pull down menu, click “PCB Decal Editor”

You’ve now created a new PCB decal. The editor automatically displays the reference designator and an additional label. You can move these labels as needed.

- Add a Terminal

- Click on the “Drafting Toolbar” button



- On the Drafting Toolbar, click on the “Terminal” button



- Set the grid to be on 0.4mm pitch by typing **g15.75** (0.4mm = 15.75mils or 0.01575”) with the mouse over the main Layout window.
- Set the display grid to be on 0.4mm pitch by typing **gd15.75** with the mouse over the main Layout window.
- Click at location 0,0 to add a terminal. The mouse’s X,Y location is given in the bottom right corner of the Layout window.
- Right click and select “Cancel” to exit Add Terminal mode.
- On the Drafting Toolbar, click the “Select” button.

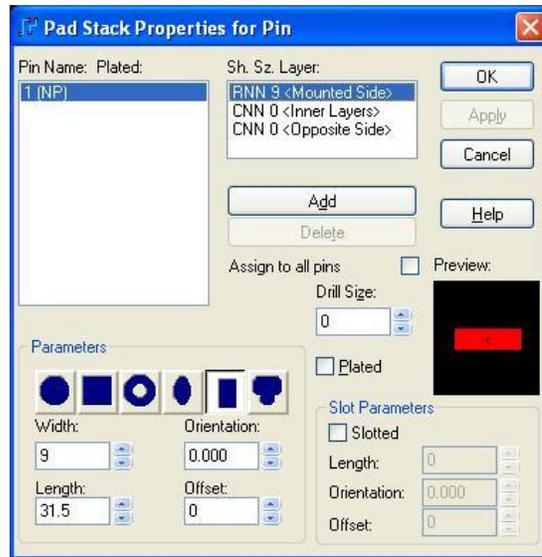


- With the mouse in the Layout window, right click and select “Select Terminals”, and then select the terminate that you just added.
- With the Terminal selected, right click and select “Pad Stacks.”

We will alter the dimensions of the Terminal in this window.

- In the “Sh. Sz. Layer” field, highlight the <Mounted Side> entry.
- In the “Parameters” field, select the Rectangular shape
- In the “Drill Size” field, enter “0” to indicate this is not a drilled pad

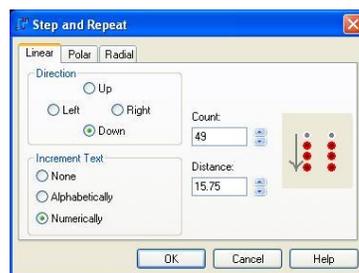
- Uncheck the “Plated” box
- In the “Parameters” field, enter “9” for the “Width”
- In the “Parameters” field, enter “31.5” for the “Length”
- Leave “Orientation” and “Offset” set to “0”.



- In the “Sh. Sz. Layer” field, highlight the <Inner Layers> entry.
- In the “Parameters” field, select the Circular shape
- In the “Parameters” field, enter “0” for the “Diameter”
- In the “Drill Size” field, enter “0”  
(Setting these values to 0 indicates that there is nothing on the Inner layers for this part)
- In the “Sh. Sz. Layer” field, highlight the <Opposite Side> entry.
- In the “Parameters” field, select the Circular shape
- In the “Parameters” field, enter “0” for the “Diameter”
- In the “Drill Size” field, enter “0”  
(Setting these values to 0 indicates that there is nothing on the Opposite layers for this part)
- Click “OK” to exit the “Pad Stack Properties” window

• Add the Remaining Terminals using “Step and Repeat”

- Select the Terminal again (right click – Select Terminal – click on the Terminal)
- Right Click and select “Step and Repeat”
  - In the “Step and Repeat” window, select the “Linear” tab
  - Choose “Down” as the Direction
  - Set the “Count” to 49
  - Set the “Distance” to 15.75
  - Set the “Increment Text” to “Numerically”
  - Click “OK” to finish the “Step and Repeat”



- Add the two Remaining Terminals manually.
  - On the Drafting Toolbar, click on the “Terminal” button



- Set the grid to be on 0.001” pitch by typing **g1** with the mouse over the main Layout window.
- Set the display grid to be on 0.001” pitch by typing **gd1** with the mouse over the main Layout window.
- Go to location **132, 48** and click to add terminal #51. The pad stack for this terminal will be the same as the previous #50 pads.
- Select terminal #51, right click, and select “Pad Stacks”.
  - In the “Parameters” field, change the “Width” value to “22”
  - In the “Parameters” field, change the “Length” value to “56”
  - Click “OK” to exit the “Pad Stacks” window
- Go to location **132, -0.820** and click to add terminal #52. The pad stack for this terminal will be the same as #51.
- Right click and select “cancel”, then click on the “Select” button on the toolbar to exit “Terminal” mode.

- Add a Component Outline

- Set the grid to 0.001” by typing **g1** with the mouse over the main Layout window.
- Set the display grid to 0.001” by typing **gd1** with the mouse over the main Layout window.



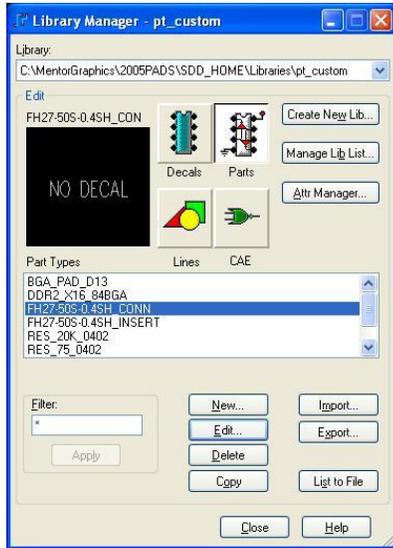
- On the “Drafting Toolbar”, select the “2D Line” button.
- With the mouse over the Layout window, right click and select “Rectangle”.
- Go to coordinate 12,59 and left click once to begin the 2D Rectangle Line.
- Go to coordinate 160,-0.831 and left click once to complete the 2D Rectangle Line.

- Save the PCB Decal

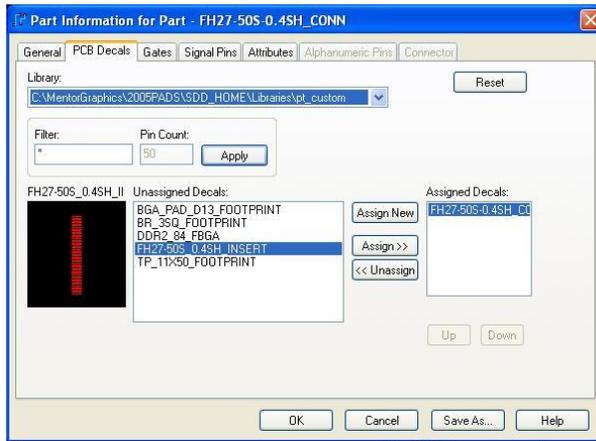
- Click the “Save” button
- Choose your custom library (ex. pt\_custom) in the Library pull down.
- Enter a name for the PCB Decal in the “Name of PCB Decal” field.  
For this example use “FH27-50S-0.4SH\_CONN\_FOOTPRINT”
- You will get a dialog asking if you want to create a new part, click “NO”

- Assign the PCB Decal to the Part

- On the “File” pull down, click “Library”,
- Choose your custom library from the “Library” pull down (ex. pt\_custom)
- Click on the “Parts” button.
- In the “Part Types” field, select “FH27-50S-0.4SH\_CONN”
- Click the “Edit” button



- In the “Part Information” window, select the “PCB Decals” tab
- Choose your custom library from the “Library” pull down (ex. pt\_custom)
- Highlight the “FH27-50S\_0.4SH\_FOOTPRINT” from the “Unassigned Decal” field and click “Assign”



- Click “OK” on the “Part Information” window.  
Now this part is complete and can be added to PADS Logic and PADS Layout.

- Exit the Decal Editor
  - On the “File” pull down menu, select “Exit Decal Editor”
- Exit PADS Layout
  - On the “File” pull down menu, select “Exit”

At this point, all parts for the design have been setup. We can now begin entering the schematic in PADS Logic.

## Step 2 : Schematic Entry

This section describes entering the schematic for this design. It consists of adding components, creating net connections, and assigning net names. In addition, design rules, stackup information, and border settings can be setup.

- Start PADS Logic

- Start → Programs → Mentor Graphics SDD → PADS2005 SP2 → System Design → PADS Logic
- At the “Welcome to PADS Logic” screen, click “Start a new design”

- Change the Sheet Boarder Size

- On the “Tools” pull down, select “Options”, and then the “Design” tab.
- In the “Sheet” section, change the “Size” field to “C”
- In the “Sheet” section, click the “Choose” button.
- Scroll down the “Drafting Items” and select “SIZEC”. Make sure that the “common” library is selected.
- Click “OK” in the “Get Drafting Item” window.
- Click “OK” in the “Options” window.

- Update the Schematic Title Blocks

- Right click and select “Select Documentation”
- Click on the “<Company Name>” text in the sheet border (bottom right)
- Right click and select “Properties”
- Enter a company name (or equivalent) and click “OK”
- Repeat this process for the other remaining schematic title blocks.

- Save the Design to a specific name

- On the “File” pull down, select “Save”
- Browse to where you wish to save this file and give a descriptive name. The file will be saved as a \*.sch file.

- Add Text Note to the Schematic

- Click on the “Add Text” button
- Enter a text note in the “Add Free Text” dialog and click “OK”.
- The text is now attached to the mouse, click where you want the text to be located.
- The “Add Free Text” dialog will appear again, click “cancel” you want the text to be located



- Add the *Hirose* Connector to the schematic

- Click on the “Add Part” button on the toolbar.



- In the “Add Part from Library” dialog, select your custom library from the “Library” pull down (ex. pt\_custom).
- Highlight the “FH27-50S-0.4SH\_CONN” part in the “Items” list. Click “Add”. The part attaches to the mouse. Click in the schematic area to place the part.
- Click “cancel” on the “Add Part” dialog to close the window.
- Hit the “Escape” button to exit the “Add Part” mode.

Notice that a **J1** Reference Designator has been automatically assigned to this part. As you add additional parts to the design, the Reference Designators will automatically increment (ex. J2, J3, ...)

- Add the *20k ohm, 0402* Resistors to the schematic

- Click on the “Add Part” button on the toolbar.
- In the “Add Part from Library” dialog, select your custom library from the “Library” pull down (ex. pt\_custom).
- Highlight the “RES\_20k\_0402” part in the “Items” list. Click “Add”. The part attaches to the mouse. Click in the schematic area to place the part.
- Click “cancel” on the “Add Part” dialog to close the window.
- Hit the “Escape” button to exit the “Add Part” mode.

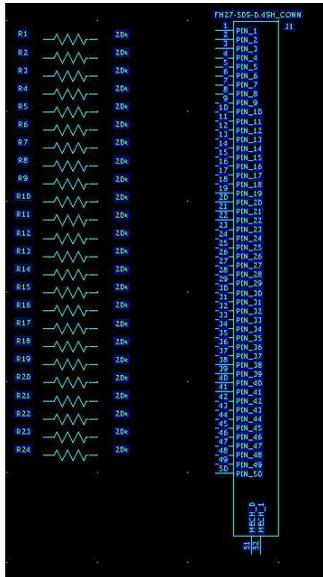
We are going add a total of 24x, “RES\_20k\_0402” resistors to this design. In order to make the schematic readable, we need to make the pin number invisible and reposition the Reference Designator and Value.

- Right Click and select “Select Parts”, select the Resistor.
- Right Click and select “Properties”
- In the “Part Properties” dialog, click the “Visibility” button.
- Uncheck the “Pin Numbers” box in the “Item Visibility” section. Click “OK” and then “Close”
- Right Click and select “Select Documentation”. Select the “R1” Reference Designator on the resistor and move it to the left of the symbol body.
- Select the “20k” Value on the resistor and move it to the right of the symbol body.

Now we can copy this resistor 23 more times in the schematic and it will have the same visibility properties.

- Right click and select “Select Part”. Select the R1 resistor.
- Copy using “CNTL-C” and Paste using “CNTL-V”. Each time you perform a Paste, the resistor attaches to the mouse. You can click to add the next resistor. Continue pasting resistors until you have a total of 24x resistors in the design. The Reference Designators will auto increment forming R1:R24.

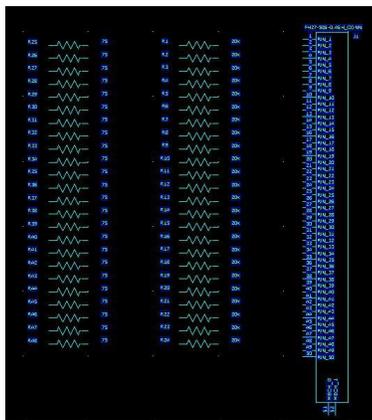
NOTE: We are going to connect the right side of the resistors to pins 2,4,6,...46,48 of Connector J1. Position the resistors so that they align with the even pins of J1.



- Add the 75ohm, 0402 Resistors to the schematic

We now want to enter 24x “RES\_20k\_0402” resistors to the design. Repeat the exact steps as in the previous section to place the 75ohm resistors. Position the 75 ohm resistors to the left of the 20k resistors so that they can easily be connected.

Reference Designators R25:R48 will automatically be assigned.



- Add signal connections between connector and resistors

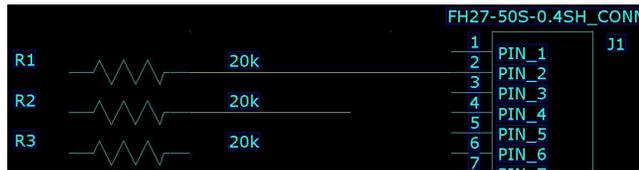
- Click on the “Add Connection” button on the toolbar.



- Click on the right node of R1 to start the connection. Click on the pin 2 node of J1 to complete the connection.

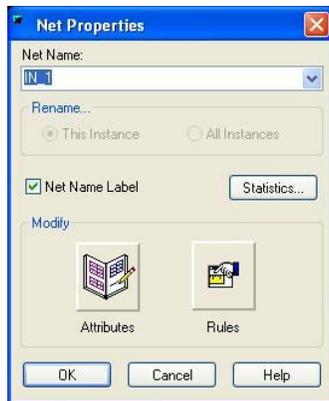
When complete, you are still in “Add Connection” mode so you can continue adding traces. When complete, you can hit escape to exit the “Add Connection” mode.

- Continue adding connections until the right sides of R1:R24 are connected to pins 2:48 on J1.



- Add Net Names to the connections

- Right Click and select “Select Connections”
- Click on the connection between R1 and J1(pin2).
- Right Click and select “Properties”
- In the “Net Properties” dialog, enter “IN\_1” for the net name.
- Check the “Net Name Label” box to make the net name visible in the schematic.
- Click “OK”.



- Now move the net name to a visible location. Right Click and select “Select Documentation”, select the net name “IN\_1” and move to the middle of the net.
- Repeat this process to add net names IN\_1:IN24 to the nets you’ve added.

- Add signal connections between resistors.

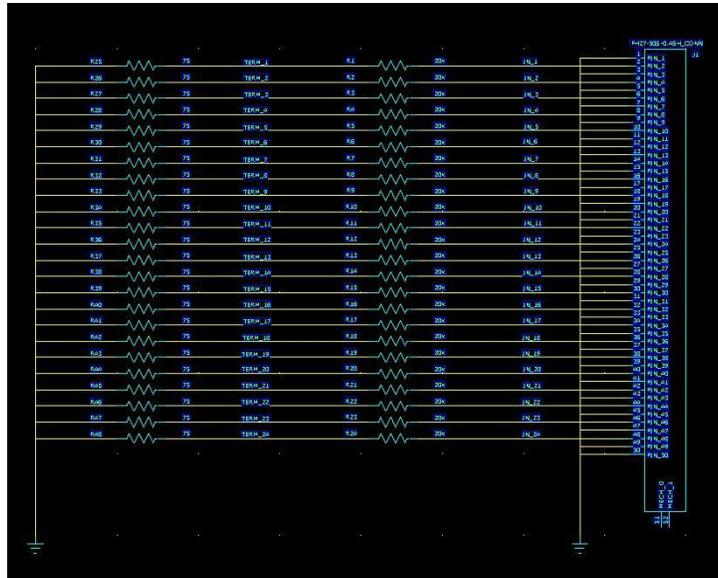
- Repeat the connection process between the left sides of R1:R24 and the right sides of R25:R48. Name the nets TERM\_0:TERM\_24.

Hint: If you move a resistor and place it so that one of its nets is directly over the net of another part, it will automatically add a net connection. You can move R25:R48 over and place them so that the right nodes overlap the left nodes of R1:R24. Then if you move R25:R48 back to their original position, 24 new net connections will be created.

- Add GND connections.

- To add a GND connection, begin a net connection by clicking on the “Add Connection” button on the toolbar.
- Before completing the net, right click and select “Ground”. This will place the GND symbol at the end of the route. Now when you click to end the connection, the GND symbol will be in your schematic. The GND symbol can be rotated by right clicking and selecting “Rotate 90”. Note that multiple pins can connect to one GND symbol. This makes the schematic less cluttered.

- Add a ground connection for all unconnected pins on connector J1. These are pins 1,3,5,...49 and pin 50.
- Add a ground connection for the left sides of R25:R48.



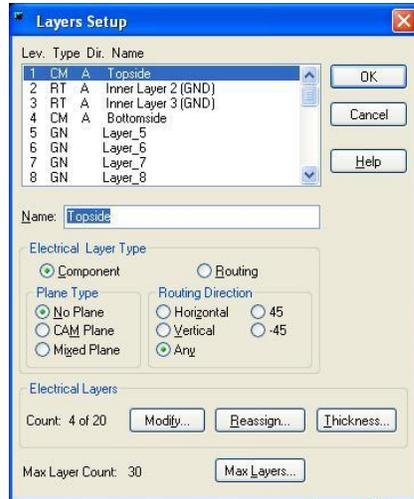
- Add “NC connections.

- Pins MECH\_0 and MECH\_1 are “No Connects”. We can add a dummy net to these pins.
- Start a new connection by clicking on the “Add Connection” button. Start the connection by clicking on the MECH\_0 pin. Before ending the net, right click and select “End”. This puts a little square on the end of the net indicating that it is no connected to anything.
- Give the net a name of “NOT\_CONNECTED” to make it apparent that you intended to make this a No Connect. This net name can be reserved for NC’s and ignored.
- Repeat for pin MECH\_1. Give a net name of “NOT\_CONNECTED”.

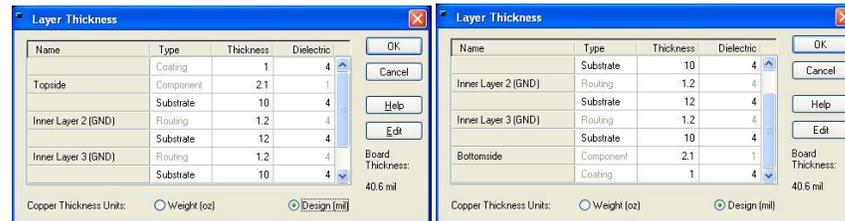
- Setup Layer Definitions

- On the “Setup” pull down, select “Layer Definition”.
  - Under “Electrical Layers”, click the “Modify” button and enter 4. Click “OK”
  - Highlight “Lev 1” in the Lev. Field.
  - change its name to “Topside” (or another descriptive name)
  - set the “Plane Type” to “No Plane”
  - set the “Routing Direction” to “Any”
  - Highlight “Lev 2” in the Lev. Field.
  - change its name to “Inner Layer 2 (GND)”
  - set the “Plane Type” to “No Plane”, we will manually add a plane to this layer.
  - set the “Routing Direction” to “Any”
  - Highlight “Lev 3” in the Lev. Field.
  - change its name to “Inner Layer 3 (GND)”
  - set the “Plane Type” to “No Plane”, we will manually add a plane to this layer.
  - set the “Routing Direction” to “Any”

- Highlight “Lev 4” in the Lev. Field.
- change its name to “Bottomside”
- set the “Plane Type” to “No Plane”, we will manually add a plane to this layer.
- set the “Routing Direction” to “Any”



- In the “Electrical Layers” section, click the “Thickness” button.
- Enter the following information:

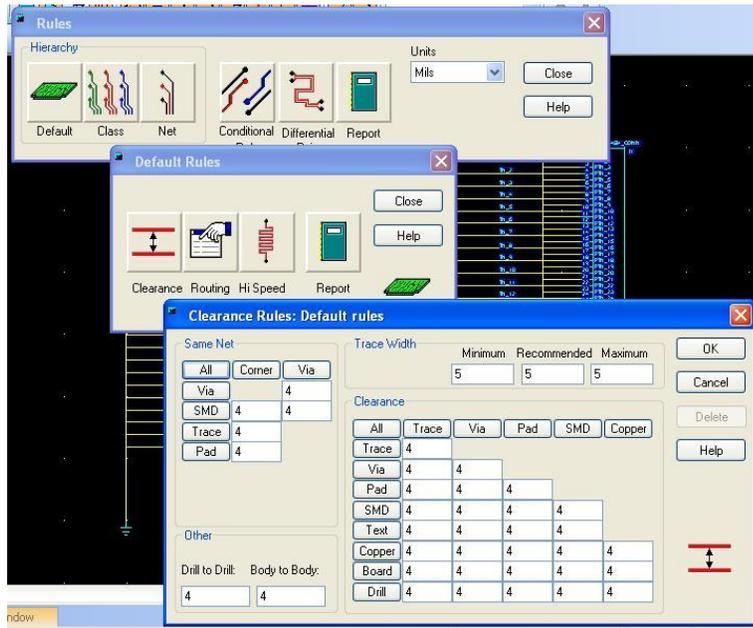


- Click “OK” to exit the “Layer Thickness” dialog.

- Click “OK” to exit the “Layer Definition” dialog.

## • Setup Design Rules

- On the “Setup” pull down, select “Design Rules”
- In the “Rules” dialog, ensure that “Mils” is set for the unit of measure.
- Click the “Default” button.
- In the “Default Rules” dialog, click “Clearance”
  - In the “Same Net” section, click the “All” button and enter “4”, click OK.
  - Enter “5” for the Minimum / Recommended / Maximum “Trace Width”
  - In the “Clearance” section, click the “All” button and enter “4”, click OK.
  - In the “Other” section, enter “4” for both “Drill-to-Drill” and “Body-to-Body”
  - Click “OK” to exit the “Clearance Rules” dialog.
- Click “Close” in the “Default Rules” dialog.
- Click “Close” in the “Rules” dialog.



At this point, the schematic is complete and we are ready to begin the physical design using PADS Layout.

### Step 3 : Layout

This section describes how to do the physical design using PADS layout. PADS Logic creates a schematic file with a \*.sch extension. PADS Layout creates a layout file with a \*.pcb extension. Design information can be passed between the two files and programs so that changes in one program can be included in the other.

To drive forward the schematic information into the layout, there is a tool called “PADS Layout Link”. Before we use this tool, we first must start a Layout and enter the board outline. Once this file exists and a board outline is present, we can use “PADS Layout Link” to pass forward the schematic data.

- Start PADS Layout

- Start → Programs → Mentor Graphics SDD → PADS2005 SP2 → PCB Layout → PADS Layout
- At the “Welcome to PADS Layout” screen, click “Start a new design”

- Give the Design a Name

- On the “File” pull down menu, click “Save”
- Give the Layout file the same name as the schematic, however the Layout file will have a &.pcb extension.

- Create a Board Outline

- Click the “Drafting Toolbar” button



We are going to enter a board outline that is 1” x “0.8”. However, we don’t want sharp corners so we will need to use the “pull arc” feature. The radius of the arch will be 0.04” at each corner. The easiest way to do this is to enter a Rectangle, then add 0.040” miters which will add diagonals to each 90 degree corner. Then we can go to each miter and convert it into an arc to give the rounded corners that we ultimately want.

- Set the grid to 0.100” by typing **g100**.
- Set the display grid to 0.100” by typing **gd100**.
- On the “Drafting Toolbar”, click on the Board Outline & Cut Out” button.



- Right Click and select “Rectangle”.
- Enter the Rectangle by clicking once at (0,0) and then again at (800,1000).
- Right Click and select “cancel”.

Now we want to make the corners into archs. First let's set the width of the Board Outline to 1.

- Right Click and select "Select Board Outline"
- Select one of the lines within the Board outline.
- Right Click and select "Select Shape"
- Right Click and select "Properties"
- Enter "1" for the "Width" and click "OK"
  
- Set the grid to 0.001" by typing **g1**.
- Set the display grid to 0.001" by typing **gd1**.
  
- Right Click and select "Select Board Outline"
- Select one of the lines within the Board outline.
- Right Click and select "Select Shape"
- Right Click and select "Add Miters"
- Enter **40** in the "Add miters" dialog and click "OK".  
This will convert all 90 degree corners into 45 degree diagonals.
  
- Right Click and select "Select Board Outline"
- Select the bottom left diagonal corner.
- Right Click and select "Pull Arc"
- Pull the arc to make a rounded corner.

Hint: If you put the mouse 1 grid point (0.001) from the straight edge of the board, it will give an ideal edge. For example, while pulling the arc, go to coordinate (39,0) and click.

- Repeat this process for the other 3 corners to make them rounded.
  
- Save and Close PADS Layout

## • Linking the Schematic to Layout

### • Start PADS Logic

- Start → Programs → Mentor Graphics SDD → PADS2005 SP2 → PCB Layout → PADS Layout

#### • Create a Netlist of the Schematic

- On the "Tools" pull down menu, select "Layout Netlist"
- Verify that the \*.asc file is going to be generated in the location that you wish.
- Make sure that every option is checked
- Click "OK"
- A text editor will pop up that shows the Netlist. Inspect this for errors and warnings.
- Verify that the Netlist is correct before proceeding.

### • Start PADS Layout from Logic

- On the "Tools" pull down menu, select "PADS Layout"
- A dialog will appear asking whether to start a NEW or OPEN an existing design, click "Open"
- Browse to the location of your \*.pcb file and click "Open"
  
- The Layout window will appear in addition to a "PADS Layout Link" window. Position the windows so that you can see the Link, Logic, and Layout.

- Send a Netlist to Layout

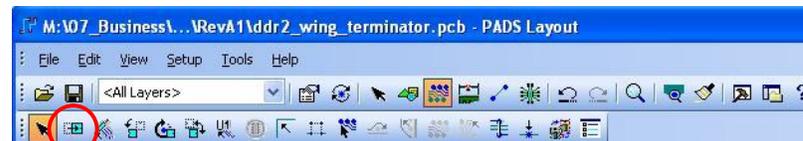
- In the “PADS Layout Link” window, click on the “Document” tab
- Verify that the path is pointing to your \*.pcb file.
- In the “PADS Layout Link” window, click on the “Preferences” tab
- Check the “Ignore Unused Pins Net” and make sure that “NOT\_CONNECTED” appears in the Name box.
- Check the “Compare...”, “Parts”, and “Nets” boxes.
  
- In the “PADS Layout Link” window, click on the “Design” tab
- Click the “Send Netlist” button.
- Now all of the parts from the schematic will appear in the Layout.

- Place the Components in Layout

- The first step is to “disperse” the components. In the Layout window, in the “Tools” pull down menu, select “Disperse Components”, click “OK” to begin disperse.
- Now if you select a component in the Logic schematic, it will be selected in the Layout window. Select J1 in the Logic window. Notice that it is selected in the Layout window. Click in the Layout window to make it active.
- Click on the “Design Toolbar” button



- Click on the “Move” button on the “Design Toolbar”



- Connector J1 is now connected to your mouse. Place the connector at (619,886) by clicking on that coordinate. You may need to set the grid to 1 and the display grid to 1.
- Continue placing the remaining 48 resistors. Keep at least 0.020” between all pads. Place the components so that they optimize the routing.



NOTE: At this point, you can close Logic if you don't want to use the LINK feature.

- Setup a Via for Routing

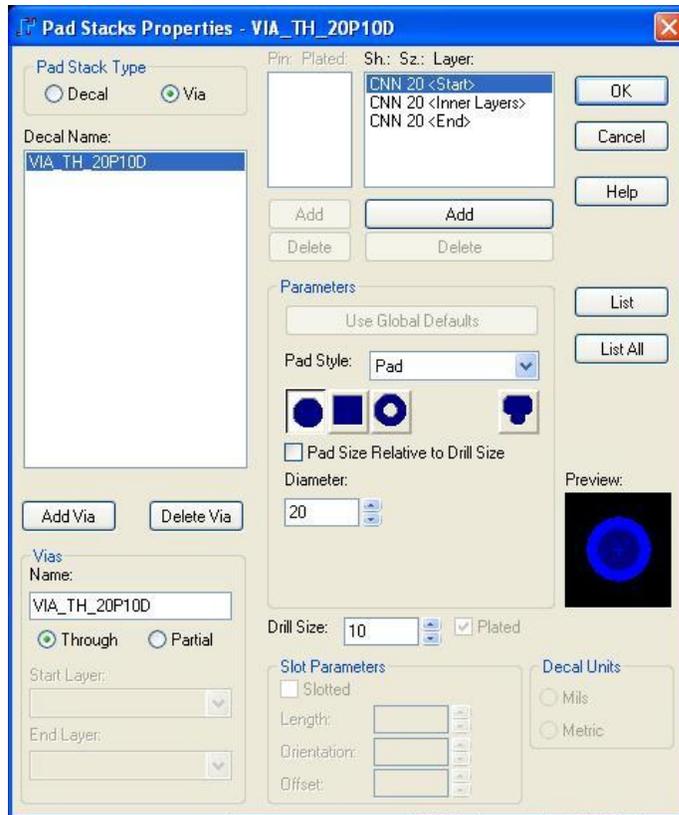
- On the layout toolbar, click the “ECO Toolbar” button. This puts you in a mode where the Pad stacks can be edited.



- In the “Options” dialog, make sure that the ECO is going to be generated in the correct location (i.e., where your design files are located). Click ”OK”

- In the “Pad Stacks Properties” Dialog, select the “Via” Type using the radio button.

- Click the “Add Via” button
- Give it a name of “VIA\_TH\_20P10D”  
(this name represents Via – Through Hole – 0.020” Pad – 0.010” Drill)
- Select “Through” in the “Vias” field
- Now Highlight the “.. <Start>” layer in the “Sh. Sz. Layer” field.
- In the “Pad Style” drop-down, select “Pad”
- Click on the Round button for the shape of the via
- Do NOT check “Pad Size Relative to Drill Size”
- Enter “20” for the “Diameter”
- Enter “10” for the “Drill Size”



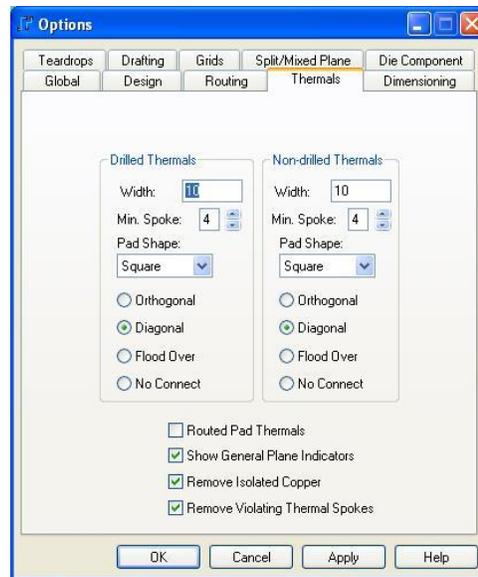
- Highlight the “.. <Inner Layers>” layer in the “Sh. Sz. Layer” field.
- Enter “20” for the “Diameter”
- Enter “10” for the “Drill Size”
  
- Highlight the “.. <End>” layer in the “Sh. Sz. Layer” field.
- Enter “20” for the “Diameter”
- Enter “10” for the “Drill Size”
  
- Highlight the “STANDARDVIA” in the “Decal Name” field.
- Click “Delete Via”

You may have to say “OK” that you will accept the PAD Stack Changes

- Click “OK” to exit the “Pad Stacks Properties” dialog.

• On the “Tools” pull down, select “Options”

- Select the “Thermals” tab
- Enter the following for both “Drilled” and “Non-drilled” thermals
  - Width = 10
  - Min Spoke = 4
  - Pad Shape = Square
  - Diagonal
  - Routed Pad Thermals = NO
  - Show General Plane Indicators = YES
  - Remove Isolated Copper = YES
  - Remove Violating Thermal Spoke= Yes



- click “OK” on the “Options” dialog

- Setup a Design Rules in Layout

- On the “Setup” pull down, select “Design Rules”
- Click on the “Default” button
- Click on the “Routing” button
- Verify that the “Selected Layers” contains the 4 layers that we setup in PADS Logic
- Verify that the “Selected Vias” contains the “VIA\_TH\_20P10D via that we just setup.
- Click “OK” to exit the “Routing” dialog
- Click “OK” to exit the “Design Rules” dialog
  
- On the “Tools” pull down, select “Options”
- Select the “Routing” tab
- In the “Layer Pair” field, choose “Topside” for the First layer and “Bottomside” for the second.
- Click “OK” to exit the “Options” dialog.

- Setup a Real Width Display

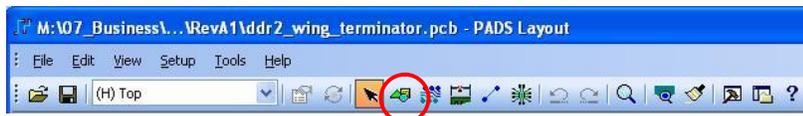
- The default setup is for Layout to only show the real width on features  $\geq 0.010$ ". Everything else will be displayed as a thin line. We want to turn on the “Real Width Feature so that the traces are shown in their actual size.

To set all lines/traces  $\geq 0.001$ " to be shown, type **r1** in the Layout window and hit Enter.

- Add the GND Planes

To route the GND nets, the nets need an ending point. Since we are manually creating the GND planes, we need to enter them first. This will allow us to route the GND pins of the connector and have an end point by dropping a via into the plane.

- Click on the “Drafting” button in the toolbar.



- Click on the “Copper Pour” button in the drafting toolbar.



- set the grid to 0.020" by typing **g20**.
- set the display grid to 0.020" by typing **gd20**.

- set the current layer to “Inner Layer 2 (GND)” by selecting it in the Layout pull down.
- Right click and select “Rectangle”
- Click at (20,20) and (780,980) to complete the rectangle.
- An “Add Drafting” dialog will appear.
- Set the “Width” to 1.
- Set the “Layer” to “Inner Layer 2 (GND)” in the pull down.
- Set the “Net” to “GND” in the pull down.
- Click “OK”

- Convert the GND Plane Corners into Rounded

- Click on the “Select” button on the drafting toolbar
- Right Click and select “Select Shapes”
- Select the Rectangle that was just added on Inner Layer 2 (GND)
- Right Click and select “Add Miters”
- Enter a Miter Radius of “40” and click “OK”
- Set the grid to **1** and “pull arcs on each miter to make into a rounded corners.

- Fill the GND Plane

- Right Click and select “Select Shapes”
- Select the GND plane outline on “Inner Layer 2 (GND)”
- Right click and select “Properties”
- Click on the “Options” button
- Uncheck the “Default” box under “Options” in the “Flood & Hatch” dialog.
- Click “OK” to exit the “Flood & Hatch” dialog.
- Click “Yes” to proceed with flood.
- Click “Cancel” to exit the “Drafting Properties” dialog

NOTE: This has flooded the GND plane and it will display the plane as “hatches”. The hatches are not real, they are just for CAD display. We can toggle between “Hatch” or “Outline Only” by typing **po** in the layout. The hatches will not show up until at least one via is dropped into or through the plane. Also, the hatching doesn’t show up all the time (maybe a bug?). So if you want a sanity check that the planes are there, you can do another “flood”, then do a **po** to see their current state.

- Repeat this process to add the same GND plane to “Inner Layer 3 (GND)”

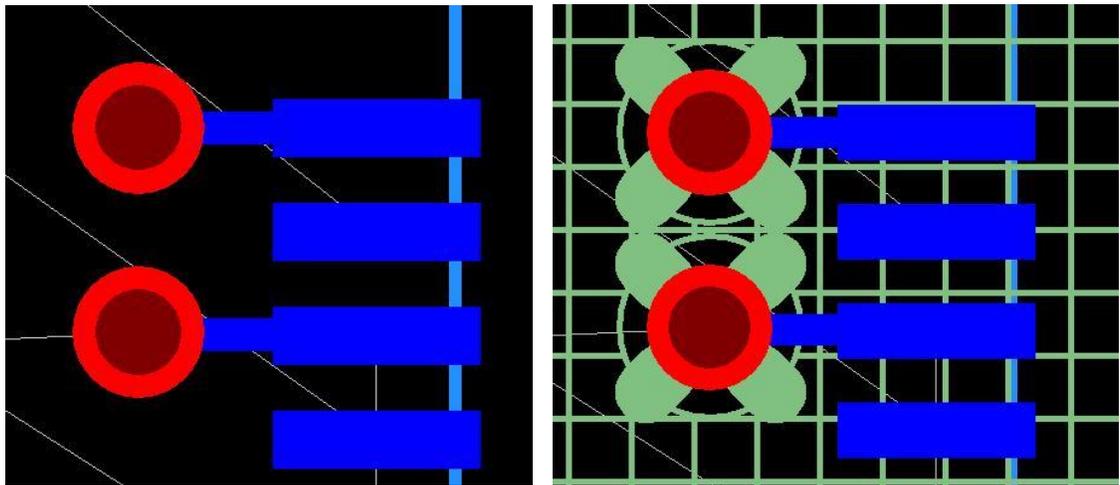
- Add the GND Connections

- Click on the “Design Toolbar”
- Choose the “Topside” layer from the pull down
- Right Click and select “Select Unroutes/Pins”
- Select J1-Pin1 to start the GND net route
- Right Click and select “Route”
- Drag the trace out to ~(583, 886) and click once.
- With the mouse over the center point (583, 886), Right Click and select “Add Via”.
- With the mouse over the center point (583, 886), Right Click and select “End”.

NOTE: This should complete the GND connection between J1-Pin1 to the GND planes on layers 2 and 3. If the “End” command doesn’t work and you are still in route mode, it is usually because your mouse wasn’t directly over the center point of the via when you clicked the right mouse button.

If that doesn’t work, you can Right Click and select “Swap End” to go to another GND pin on J1. If you add another via, then usually the “End” command will terminate the route.

After adding a couple of GND vias, you can turn on **po** and check the ground plane hatching and thermals. You might need to perform another “Flood” command.



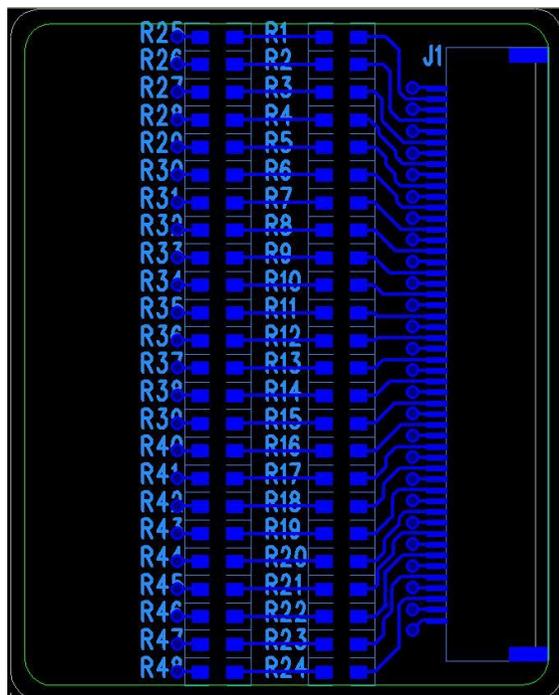
- Route the remaining GND connections.

- Route the Signal Traces

- Click on the “Design” button in the toolbar.
- Right Click and select “Select Unroutes/Pins”
- Select the ‘guide’ line coming out of J1-Pin2
- Right Click and select “Route”

Some useful commands are:

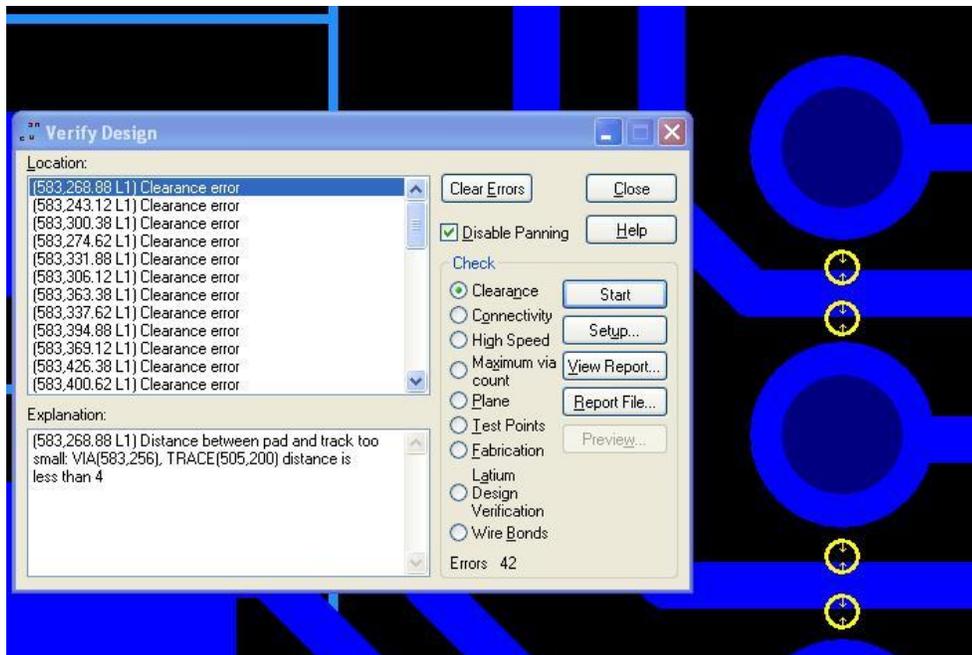
- Right Click and select “Swap End”, this changes your selection to the other end of the net.
  - Right Click and select “Complete”, this automatically finishes the connection.
- Complete the Route for net “IN\_1”. The net name is shown in the lower left corner of the Layout window.
  - Complete the Routes for all of the signal nets in the design.



## Step 4 : Design Rule Check (DRC)

This section describes checking the design for routing and placement errors. The DRC operates on what part of the design is visible on the screen. This provides a way to narrow in on a certain area and perform local DRC's. Make sure that the final DRC is done with the entire board visible.

- On the “Tools” menu, select “Verify Design”
  - Check the “Clearance” radio button to perform a clearance DRC.
  - Click the “Setup” button.
  - Make sure the “Net to All”, “Board Outline”, and “Drill to Drill” options are checked. Select other options that you want to check. Click “OK” to accept close the dialog.
  - Click “Start”. Any errors that were found will be listed and highlighted in the layout.



- The rules being checked are what were setup in the “Default” routing rules prior to doing layout. If we wish to allow a certain error, we can alter the routing rules. For example, in this design, the only 0.004” violations were when the traces went through the Vias. We could either change the rules to look for 0.003”, or reduce the trace width for this segment of this route.
- Click on the “Connectivity” radio button and click “Start”
- Fix all errors before proceeding.

## Step 5 : Generating Fabrication Files

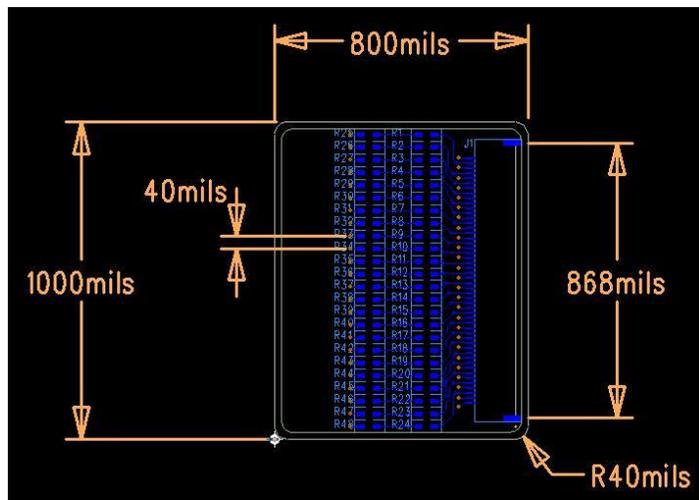
This section describes how to create the Computer Aided Manufacturing (CAM) files that will be sent to the fab shop to build this printed circuit board. The following files will need to be generated for this 4-layer board:

<u>Combined Fab and Drill Drawing</u>	This is a drawing that contains dimensional information about the PCB. It also contains stackup and construction information. Also a Drill Table is included. CBDWG_NDRAM-26401.pdf <b>File Name (s) : CBDWG_boardnum.pdf</b>
<u>NC Drill Data</u>	This is a file that contains the XY coordinates for all of the drill holes in the design. <b>File Name (s) : drill.drl, drill.lst, drill.rep</b>
<u>Top Silkscreen</u>	This file defines the text and shapes that are to be added to the outer layer of the PCB for purposes of documentation. <b>File Name (s) : top_silk.pho, top_silk.rep</b>
<u>Top Solder Mask</u>	This file defines the regions of copper that need to be covered or exposed with Solder Mask. <b>File Name (s) : top_mask.pho, top_mask.rep</b>
<u>Top Solder Paste</u>	This file is used by the load shop and defines the regions where Solder Paste will be placed so that components can be loaded onto the board. <b>File Name (s) : top_paste.pho, top_paste.rep</b>
<u>Topside Artwork</u>	This is a photo plot file, (also called GERBER) that is used to “print” the copper features for this layer. <b>File Name (s) : top.pho, top.rep</b>
<u>Layer 2 Artwork</u>	This is a photo plot file, (also called GERBER) that is used to “print” the copper features for this layer. <b>File Name (s) : L2.pho, L2.rep</b>
<u>Layer 3 Artwork</u>	This is a photo plot file, (also called GERBER) that is used to “print” the copper features for this layer. <b>File Name (s) : L3.pho, L3.rep</b>
<u>Bottomside Artwork</u>	This is a photo plot file, (also called GERBER) that is used to “print” the copper features for this layer. <b>File Name (s) : bottom.pho, bottom.rep</b>
<u>Bottom Silkscreen</u>	This file defines the text and shapes that are to be added to the outer layer of the PCB for purposes of documentation. <b>File Name (s) : bottom_silk.pho, bottom_silk.rep</b>
<u>Bottom Solder Mask</u>	This file defines the regions of copper that need to be covered exposed with Solder Mask. <b>File Name (s) : bottom_mask.pho, bottom_mask.rep</b>
<u>Bottom Solder Paste</u>	This file is used by the load shop and defines the regions where Solder Paste will be placed so that components can be loaded onto the board. <b>File Name (s) : bottom_paste.pho, bottom_paste.rep</b>
<u>Aperture Report</u>	This file contains the geometries needed to fabricate the PCB. <b>File Name (s) : aperture_report.rep</b>

- Create the Combined Fab and Drill Drawing (CBDWG\_boardnum.pdf)

We will insert a sheet border on the “Drill Drawing” layer of the Layout design. We can then add dimensions and notes for the fab shop. We will then have the tool generate a drill table for us that will show up on our drawing when we print it.

- Turn on the “Drill Drawing” layer (CNTL-ALT-C)
- Add a Sheet border
  - Click on the “Drafting Toolbar” button.
  - Click on the “From Library” button.
  - Select a Sheet border from the list.  
The size should be such that the PCB dimensions can be easily seen.
  - Fill out the sheet border information.
  - Add Notes for the fabricator.
  - Add a Stackup drawing for the fabricator.
- Add Dimensions
  - On the “Tools” pull down, select “Options”
  - Select the “Dimensioning” tab
  - Select “General Settings” from the pull down.
  - Set the “Text” and “Lines” to the “Drill Drawing” layer
  - Click “OK” to save these settings
  - Click on the “Dimensioning” button on the toolbar.
  - Right click and select “Select Board Outline”
  - On the “Dimensioning Toolbar”, click the “Horizontal” button
  - Right click and select “Any Point”
  - Now select the left side of the Board Outline to enter the first point.
  - Now select the right side of the Board Outline to enter the second point.
  - Position the Dimension above the Board Outline
  - Add other dimensions that are critical to understanding the board construction.

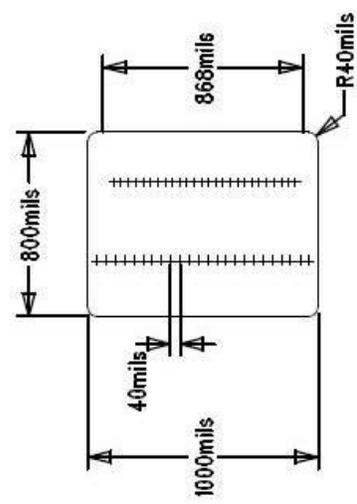
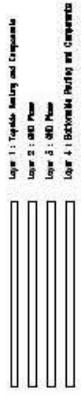


- Add Drill Table
  - Turn off all layers except “Drill Drawing”
  - On the “File” pull down, select “CAM
  - Highlight all of the default CAM documents and click “Delete”
  - Manually create a directory named “CAM” in your design directory.
  - In the “CAM Directory” field, select “<Create>
  - Browse to your design directory and create a directory called “CAM”
  - Click “OK”.
  
  - Click the “Add” button
  - In the “Document Name” field, enter “Drill Drawing”
  - In the “Document Type”, select “Drill Drawing”
    - In the “Layer Association” popup, select “Topside”
  - In the “Output File” field, enter “drill.pho”
  
  - Click the “Set Layers” button.
  - Select “Drill Drawing”.
  - Make sure the “Drill Drawing” radio button is checked.
  - Click “OK”
  
  - Click the “Layers” button.
  - In the “selected” field, remove all other layers except for “Drill Drawing”.
  - Highlight “Drill Drawing”. Check the “Items on Primary” boxes for “vias”
  - Click “OK”
  
  - Click the “Options” button.
  - In the “Justification” field, select “Scale to Fit”
  - Uncheck the “Plot Job Name” option.
  - Click the “Drill Symbols” button.
    - In the “Location – Y” field, enter “-750” to offset the drill table.
    - Click “OK”
  - Click “OK”
  
  - Click the “Print” button.
  - Click the “Device Setup” button.
  - Select a PDF output device and “Landscape” Orientation, Click “OK”
  
  - Click the “Preview Selection”
  - Make sure everything looks the way you want and that the text is readable.
  - Click “Close”
  - Click “OK” to return to the “Define CAM Documents” dialog
  
  - In the “Define CAM Documents” dialog, highlight “Drill Drawing”
  - Click the “Run” button to generate the PDF drawing in your CAM directory.
  - You will need to specify the path and name of the file.
  - The following is an example of a combined fab and drill drawing.
  - Click “Save” to save this artwork document in your CAM file list

6 5 4 3 2 1

**PTI RESTRICTED**

THIS DOCUMENT AND ANY ASSOCIATED DATA CONTAIN RESTRICTED INFORMATION THAT IS PTI PROPERTY. ONLY DISCLOSE OR DUPLICATE FOR OTHERS AS AUTHORIZED BY PTI.



SIZE	QTY	SYM	PLATED	TOL
0.01	50	+	YES	+/-0.0

DESIGNATION	REV	DATE	BY
NDRAM-26407-1			
DESIGNER	APPROVER	DATE	BY
Brock J. Lomeres		11-1-06	

ITEM	QTY	DESCRIPTION	UNIT	REVISION
1	50	PRINTED CIRCUIT BOARD	PCB	
2	50	TOPSIDE SOLDER	MG	
3	50	400 PLEAD	MG	
4	50	400 PLEAD	MG	
5	50	BOTTOMSIDE PREPREG	MG	
6	50	COMPONENTS	PCS	

1. FINISH ON FACE OF BOARD: 61A
2. PCB DESIGNED FOR 11--00.
3. 0.044" THICK REQUIRED IN THE OTHER BLENDS SPECIFIED OTHERWISE.
4. BOARD IS TO BE UNILAYER REPAIRED BY PERMIT BUREAU (CONTRACT, STATEMENT, OR IN APPROVED CONTRACT).
5. DIMENSIONS AND TOLERANCES ARE SHOWN IN THIS DRAWING (MIL).
6. ALL DIMENSIONS ARE TO CENTER UNLESS OTHERWISE SPECIFIED.
7. ALL DIMENSIONS ARE TO CENTER UNLESS OTHERWISE SPECIFIED.
8. HOLE AND SLOT DIMENSIONS MAY BE TOLERANCES FOR DIMENSIONS.
9. HOLE AND SLOT DIMENSIONS MAY BE TOLERANCES FOR DIMENSIONS.
10. HOLE AND SLOT DIMENSIONS MAY BE TOLERANCES FOR DIMENSIONS.
11. HOLE AND SLOT DIMENSIONS MAY BE TOLERANCES FOR DIMENSIONS.
12. HOLE AND SLOT DIMENSIONS MAY BE TOLERANCES FOR DIMENSIONS.
13. HOLE AND SLOT DIMENSIONS MAY BE TOLERANCES FOR DIMENSIONS.
14. HOLE AND SLOT DIMENSIONS MAY BE TOLERANCES FOR DIMENSIONS.
15. HOLE AND SLOT DIMENSIONS MAY BE TOLERANCES FOR DIMENSIONS.
16. HOLE AND SLOT DIMENSIONS MAY BE TOLERANCES FOR DIMENSIONS.
17. HOLE AND SLOT DIMENSIONS MAY BE TOLERANCES FOR DIMENSIONS.
18. HOLE AND SLOT DIMENSIONS MAY BE TOLERANCES FOR DIMENSIONS.
19. HOLE AND SLOT DIMENSIONS MAY BE TOLERANCES FOR DIMENSIONS.
20. HOLE AND SLOT DIMENSIONS MAY BE TOLERANCES FOR DIMENSIONS.

DO NOT SCALE THIS DRAWING

DESIGNER	APPROVER	DATE	BY
Brock J. Lomeres		11-1-06	
DESIGNATION	REV	DATE	BY
NDRAM-26407-1			
DESIGNER	APPROVER	DATE	BY
Brock J. Lomeres		11-1-06	
DESIGNATION	REV	DATE	BY
NDRAM-26407-1			

- Create the NC Drill Data Files (**drill.drl**, **drill.lst**, **drill.rep**)

An NC Drill Data file is an XY table of where drill locations will be. If using a combination of TH, Blind, or Buried Vias, each drill stage would need an individual NC drill file. For this example, we are using one type of via that requires one drill size. This means we need to only setup one NC Drill output. The CAM output will automatically create the three files (drill.drl, drill.lst, drill.rep).

- On the “File” pull down, select “CAM” (if not in the CAM mode already)
  - Click the “Add” button.
  - Enter “NC Drill Data” for the “Document” name.
  - Select “NC Drill” for the “Document Type”.
  - Enter “drill.drl” for the “Output File”.
  - Leave the “Fabrication Layer” “Unassigned”.
  - Click the “Options” button.
  - Set the “Origin Offset” to X=0, Y=0.
  - Make sure “Through Vias”, “Plated Pins”, and “Non-Plated Pins” are checked.
  - Click “OK”
  - Click “OK”
- In the “Define CAM Documents” dialog, highlight “NC Data”
- Click the “Run” button to generate the NC Drill data in your CAM directory.
- If you go to your CAM directory and open drill.lst in a text editor, you will see a list of XY coordinates. Verify that the coordinates make sense.

```
Drill Listing
=====
Drill: .01 Tool: 1 Feed: 0 Speed: 0
X 24200 Y 4000
X 24200 Y 8000
X 24200 Y 12000
```

- If you go to your CAM directory and open drill.drl in a text editor, you will see a list of XY coordinates in a slightly different format. Verify that the coordinates make sense.

```
%
T1C.01F0S0
X00242Y0004
X00242Y0008
X00242Y0012
```

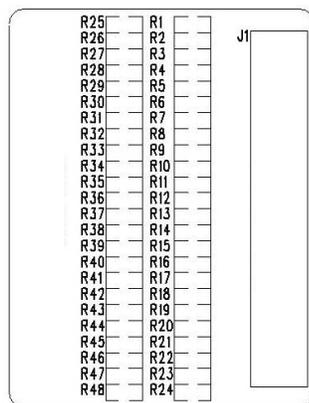
- If you go to your CAM directory and open drill.rep in a text editor, you will see table that summarizes the drill data. Verify every looks OK.

```
Drill Sizes Report
=====
Tool Size Pltd Feed Speed Qty
====  =====  =====  =====  =====  =====
1 10 x 0 0 50
```

- Click “Save” to save this artwork document in your CAM file list.

- Create the Top Silkscreen Artwork (**top\_silk.pho**, **top\_silk.rep**)
  - On the “File” pull down, select “CAM” (if not in the CAM mode already)
    - Click the “Add” button.
    - Enter “Top Silkscreen” for the “Document” name.
    - Select “Silkscreen” for the “Document Type”.
    - Select “Topside” in the “Layer Association” popup, click “OK”
    - Enter “top\_silk.pho” for the “Output File”.
    - Leave the “Fabrication Layer” “Silkscreen Top”.
    - Click on the “Layers” button
    - In the “Selected” field, only the “Silkscreen Top” layer should be present
    - Highlight “Silkscreen Top”
      - In the “Other” field, check “Board Outline”
      - In “Items on Primary”, check “Lines”, “Ref. Des”, “Text”, and “Outlines”
      - In “Component outlines”, check “Top Mounted”
    - Click “OK” on the “Select Items” dialog.
    - Click the “Options” button.
    - Set the Orientation to “0” and “Justification” to “Bottom Left”.
    - Set the “Scaling” to “1 to 1” and the “Offset” to X=0, Y=0.
    - Click “OK” on the “Plot Options” dialog.
    - Click “OK” on the “Edit Document” dialog.

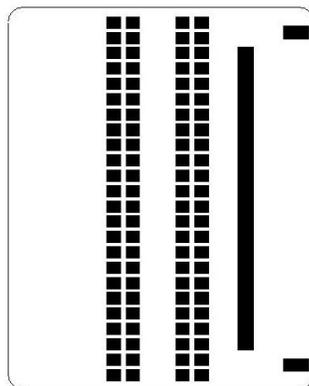
At this point, you can preview the Silkscreen settings. Highlight the “Top Silkscreen” document and click the “Preview” button. You should see something like the following: If unwanted text is present or missing, you will need to go into Layout, enter the “ECO” mode, select the part, and then “Edit Decal”. In the Part Editor, you can reassign text to different layers. Once the Silkscreen looks OK, you can generate the artwork files.



- Highlight the “Top Silkscreen” document and click “Run”.
- Verify that the files were created in your CAM directory.
- Click the “Save” button in the “Define CAM Documents” dialog.

- Create the Top Solder Mask Artwork (**top\_mask.pho**, **top\_mask.rep**)
  - On the “File” pull down, select “CAM” (if not in the CAM mode already)
    - Click the “Add” button.
    - Enter “Top Solder Mask” for the “Document” name.
    - Select “Solder Mask” for the “Document Type”.
    - Select “Topside” in the “Layer Association” popup, click “OK”
    - Enter “top\_mask.pho” for the “Output File”.
    - Leave the “Fabrication Layer” “Solder Mask Top”.
    - Click on the “Layers” button
    - In the “Selected” field, the “Topside” and “Silkscreen Top” should be present.
    - Highlight “Topside”
    - In the “Other” field, check “Board Outline”
    - Click “OK” in the “Select Items” dialog.
    - Click the ”Options” button.
    - Set the Orientation to “0” and “Justification” to “Bottom Left”
    - Set the “Scaling” to “1 to 1” and the “Offset” to X=0, Y=0
    - Click “OK” on the “Plot Options” dialog.
    - Click “OK” on the “Edit Document” dialog.

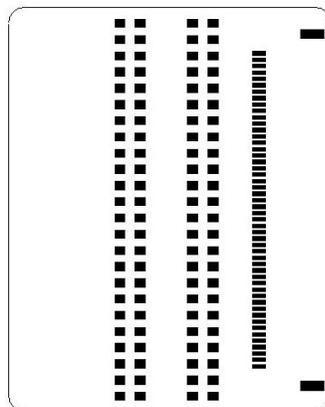
At this point, you can preview the document settings. Highlight the “Top Solder Mask” document and click the “Preview” button. You should see something like the following: This image is negative, which means Solder Mask will exist everywhere except indicated. All of the pads should be exposed. The connector is too fine of a pitch for solder mask to be placed between the pads. As a result, there is one mask opening for all of the pads. If the Solder Mask is not correct, you will need to go into Layout, enter the “ECO” mode, select the part, and then “Edit Decal”. In the Part Editor, you can manually add Mask. Once the Solder Mask looks OK, you can generate the artwork files.



- Highlight the “Top Solder Mask” document and click “Run”.
- Verify that the files were created in your CAM directory.
- Click the “Save” button in the ”Define CAM Documents” dialog.

- Create the Top Solder Paste Artwork (**top\_paste.pho**, **top\_paste.rep**)
  - On the “File” pull down, select “CAM” (if not in the CAM mode already)
    - Click the “Add” button.
    - Enter “Top Solder Paste” for the “Document” name.
    - Select “Paste Mask” for the “Document Type”.
    - Select “Topside” in the “Layer Association” popup, click “OK”
    - Enter “top\_paste.pho” for the “Output File”.
    - Leave the “Fabrication Layer” “Paste Mask Top”.
    - Click on the “Layers” button
    - In the “Selected” field, the “Topside” and “Paste Mask Top” should be present.
    - Highlight “Topside”
    - In the “Other” field, check “Board Outline”
    - Click “OK” in the “Select Items” dialog.
    - Click the “Options” button.
    - Set the Orientation to “0” and “Justification” to “Bottom Left”
    - Set the “Scaling” to “1 to 1” and the “Offset” to X=0, Y=0
    - Click “OK” on the “Plot Options” dialog.
    - Click “OK” on the “Edit Document” dialog.

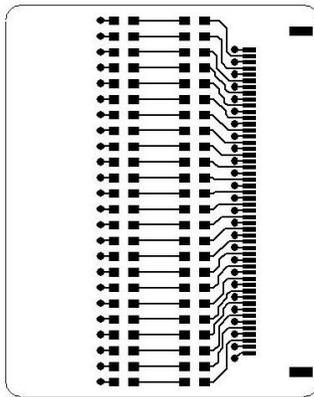
At this point, you can preview the document settings. Highlight the document and click the “Preview” button. You should see something like the following: This image is negative, which means Solder Paste will exist everywhere except indicated. All of the pads should have some paste. The *Hirose* connector has special paste requirements, consult the datasheet to manually alter the paste mask. As a result, there is one mask opening for all of the pads. If the Solder Paste is not correct, you will need to go into Layout, enter the “ECO” mode, select the part, and then “Edit Decal”. In the Part Editor, you can manually add Paste. Once the Solder Paste looks OK, you can generate the artwork files.



- Highlight the “Top Solder Paste” document and click “Run”.
- Verify that the files were created in your CAM directory.
- Click the “Save” button in the “Define CAM Documents” dialog.

- Create the Topside Metal Artwork (**top.pho**, **top.rep**)
  - On the “File” pull down, select “CAM” (if not in the CAM mode already)
    - Click the “Add” button.
    - Enter “Topside Artwork” for the “Document” name.
    - Select “Routing/Split Plane” for the “Document Type”.
    - Select “Topside” in the “Layer Association” popup, click “OK”
    - Enter “top.pho” for the “Output File”.
    - Leave the “Fabrication Layer” “Topside”.
    - Click on the “Layers” button
    - In the “Selected” field, “Topside” should be present.
    - Highlight “Topside”
    - In the “Other” field, check “Board Outline”
    - Click “OK” in the “Select Items” dialog.
    - Click the “Options” button.
    - Set the Orientation to “0” and “Justification” to “Bottom Left”
    - Set the “Scaling” to “1 to 1” and the “Offset” to X=0, Y=0
    - Click “OK” on the “Plot Options” dialog.
    - Click “OK” on the “Edit Document” dialog.

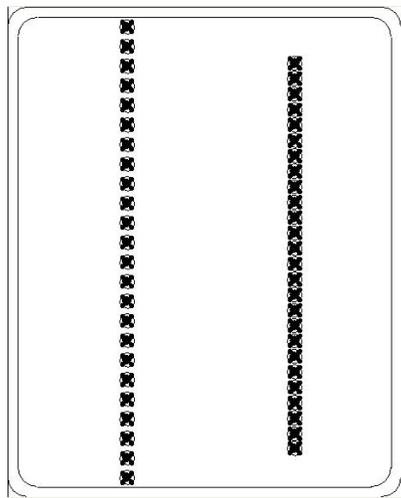
At this point, you can preview the document settings. Highlight the document and click the “Preview” button. You should see something like the following: Once the Artwork looks OK, you can generate the artwork files.



- Highlight the “Topside Artwork” document and click “Run”.
- Verify that the files were created in your CAM directory.
- Click the “Save” button in the “Define CAM Documents” dialog.

- Create the Layer 2 Metal Artwork (**L2.pho, L2.rep**)
  - On the “File” pull down, select “CAM” (if not in the CAM mode already)
    - Click the “Add” button.
    - Enter “L2 Artwork” for the “Document” name.
    - Select “Routing/Split Plane” for the “Document Type”.
    - Select “Inner Layer 2 (GND)” in the “Layer Association” popup, click “OK”
    - Enter “L2.pho” for the “Output File”.
    - Leave the “Fabrication Layer” “Inner Layer 2 (GND)”.
    - Click on the “Layers” button
    - In the “Selected” field, “Inner Layer 2 (GND)” should be present.
    - Highlight “Inner Layer 2 (GND)”
    - In the “Other” field, check “Board Outline”
    - Click “OK” in the “Select Items” dialog.
    - Click the “Options” button.
    - Set the Orientation to “0” and “Justification” to “Bottom Left”
    - Set the “Scaling” to “1 to 1” and the “Offset” to X=0, Y=0
    - Click “OK” in the “Add Document” dialog.

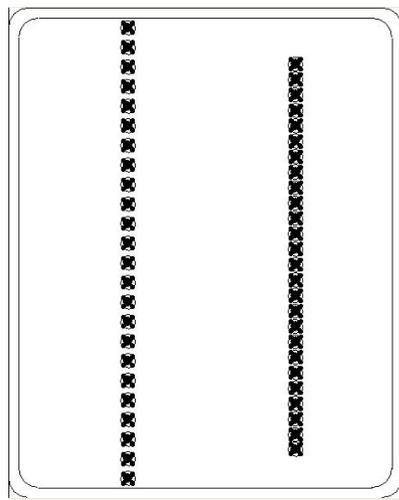
At this point, you can preview the document settings. Highlight the document and click the “Preview” button. You should see something like the following:  
Once the Artwork looks OK, you can generate the artwork files.



- Highlight the “L2 Artwork” document and click “Run”.
- Verify that the files were created in your CAM directory.
- Click the “Save” button in the “Define CAM Documents” dialog.

- Create the Layer 3 Metal Artwork (**L3.pho**, **L3.rep**)
  - On the “File” pull down, select “CAM” (if not in the CAM mode already)
    - Click the “Add” button.
    - Enter “L3 Artwork” for the “Document” name.
    - Select “Routing/Split Plane” for the “Document Type”.
    - Select “Inner Layer 3 (GND)” in the “Layer Association” popup, click “OK”
    - Enter “L3.pho” for the “Output File”.
    - Leave the “Fabrication Layer” “Inner Layer 3 (GND)”.
    - Click on the “Layers” button
    - In the “Selected” field, “Inner Layer 3 (GND)” should be present.
    - Highlight “Inner Layer 3 (GND)”
    - In the “Other” field, check “Board Outline”
    - Click “OK” in the “Select Items” dialog.
    - Click the “Options” button.
    - Set the Orientation to “0” and “Justification” to “Bottom Left”
    - Set the “Scaling” to “1 to 1” and the “Offset” to X=0, Y=0
    - Click “OK” in the “Add Document” dialog.

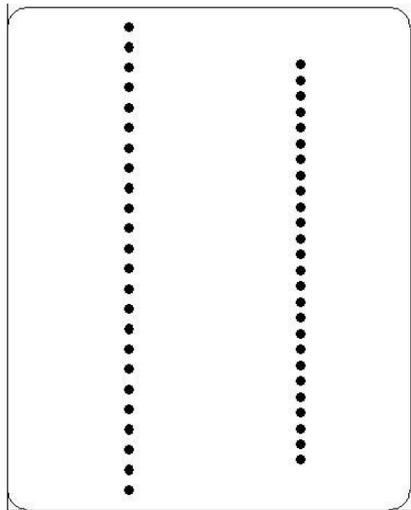
At this point, you can preview the document settings. Highlight the document and click the “Preview” button. You should see something like the following:  
Once the Artwork looks OK, you can generate the artwork files.



- Highlight the “Bottomside Artwork” document and click “Run”.
- Verify that the files were created in your CAM directory.
- Click the “Save” button in the “Define CAM Documents” dialog.

- Create the Bottomside Metal Artwork (**bottom.pho, bottom.rep**)
  - On the “File” pull down, select “CAM” (if not in the CAM mode already)
    - Click the “Add” button.
    - Enter “Bottomside Artwork” for the “Document” name.
    - Select “Routing/Split Plane” for the “Document Type”.
    - Select “Bottomside” in the “Layer Association” popup, click “OK”
    - Enter “bottom.pho” for the “Output File”.
    - Leave the “Fabrication Layer” “Bottomside”.
    - Click on the “Layers” button
    - In the “Selected” field, “Bottomside” should be present.
    - Highlight “Bottomside”
    - In the “Other” field, check “Board Outline”
    - Click “OK” in the “Select Items” dialog.
    - Click the “Options” button.
    - Set the Orientation to “0” and “Justification” to “Bottom Left”
    - Set the “Scaling” to “1 to 1” and the “Offset” to X=0, Y=0
    - Click “OK” on the “Plot Options” dialog.
    - Click “OK” on the “Edit Document” dialog.

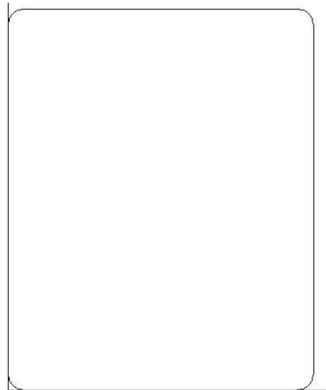
At this point, you can preview the document settings. Highlight the document and click the “Preview” button. You should see something like the following: Once the Artwork looks OK, you can generate the artwork files.



- Highlight the “Bottomside Artwork” document and click “Run”.
- Verify that the files were created in your CAM directory.
- Click the “Save” button in the “Define CAM Documents” dialog.

- Create the Bottomside Silkscreen Artwork (**bottom\_silk.pho**, **bottom\_silk.rep**)
  - On the “File” pull down, select “CAM” (if not in the CAM mode already)
    - Click the “Add” button.
    - Enter “Bottom Silkscreen” for the “Document” name.
    - Select “Silkscreen” for the “Document Type”.
    - Select “Bottomside” in the “Layer Association” popup, click “OK”
    - Enter “bottom\_silk.pho” for the “Output File”.
    - Leave the “Fabrication Layer” “Silkscreen Bottom”.
    - Click on the “Layers” button
    - In the “Selected” field, only the “Silkscreen Bottom” layer should be present
    - Highlight “Silkscreen Bottom”
      - In the “Other” field, check “Board Outline”
      - In “Items on Primary”, check “Lines”, “Ref. Des”, “Text”, and “Outlines”
      - In “Component outlines”, check “Bottomside Mounted”
    - Click “OK” on the “Select Items” dialog.
    - Click the “Options” button.
    - Set the Orientation to “0” and “Justification” to “Bottom Left”.
    - Set the “Scaling” to “1 to 1” and the “Offset” to X=0, Y=0.
    - Click “OK” on the “Plot Options” dialog.
    - Click “OK” on the “Edit Document” dialog.

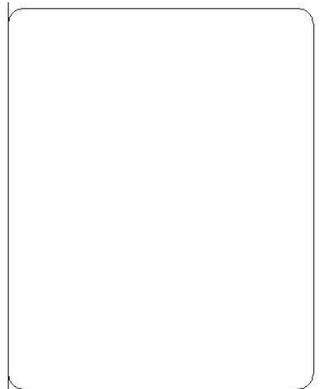
At this point, you can preview the Silkscreen settings. Highlight the “Bottom Silkscreen” document and click the “Preview” button. For this design, you should see a blank outline because we did not place any silkscreen on the bottom side. Once the Silkscreen looks OK, you can generate the artwork files.



- Highlight the “Bottom Silkscreen” document and click “Run”.
- Verify that the files were created in your CAM directory.
- Click the “Save” button in the “Define CAM Documents” dialog.

- Create the Bottom Solder Mask Artwork (**bottom\_mask.pho, bottom\_mask.rep**)
  - On the “File” pull down, select “CAM” (if not in the CAM mode already)
    - Click the “Add” button.
    - Enter “Bottom Solder Mask” for the “Document” name.
    - Select “Solder Mask” for the “Document Type”.
    - Select “Bottomside” in the “Layer Association” popup, click “OK”
    - Enter “bottom\_mask.pho” for the “Output File”.
    - Leave the “Fabrication Layer” “Solder Mask Bottom”.
    - Click on the “Layers” button
    - In the “Selected” field, the “Bottomside” and “Solder Mask Bottom” exist.
    - Highlight “Bottomside”
    - In the “Other” field, check “Board Outline”
    - Click “OK” in the “Select Items” dialog.
    - Click the ”Options” button.
    - Set the Orientation to “0” and “Justification” to “Bottom Left”
    - Set the “Scaling” to “1 to 1” and the “Offset” to X=0, Y=0
    - Click “OK” on the “Plot Options” dialog.
    - Click “OK” on the “Edit Document” dialog.

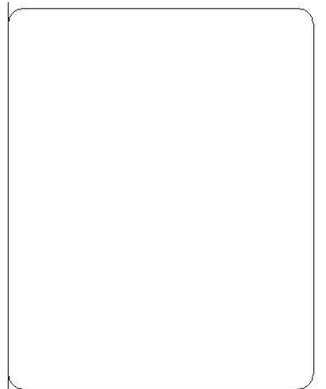
At this point, you can preview the Mask settings. Highlight the “Bottom Solder Mask” document and click the “Preview” button. For this design, you should see a blank outline because we did not place any components on the bottom side. Once the Mask looks OK, you can generate the artwork files.



- Highlight the “Bottom Solder Mask” document and click “Run”.
- Verify that the files were created in your CAM directory.
- Click the “Save” button in the ”Define CAM Documents” dialog.

- Create the Bottom Solder Paste Artwork (**bottom\_paste.pho, bottom\_paste.rep**)
  - On the “File” pull down, select “CAM” (if not in the CAM mode already)
    - Click the “Add” button.
    - Enter “Bottom Solder Paste” for the “Document” name.
    - Select “Paste Mask” for the “Document Type”.
    - Select “Bottomside” in the “Layer Association” popup, click “OK”
    - Enter “bottom\_paste.pho” for the “Output File”.
    - Leave the “Fabrication Layer” “Paste Mask Bottom”.
    - Click on the “Layers” button
    - In the “Selected” field, the “Bottomside” and “Paste Mask Bottom” exist.
    - Highlight “Bottomside”
    - In the “Other” field, check “Board Outline”
    - Click “OK” in the “Select Items” dialog.
    - Click the “Options” button.
    - Set the Orientation to “0” and “Justification” to “Bottom Left”
    - Set the “Scaling” to “1 to 1” and the “Offset” to X=0, Y=0
    - Click “OK” on the “Plot Options” dialog.
    - Click “OK” on the “Edit Document” dialog.

At this point, you can preview the Paste settings. Highlight the “Bottom Solder Paste” document and click the “Preview” button. For this design, you should see a blank outline because we did not place any components on the bottom side. Once the Paste looks OK, you can generate the artwork files.



- Highlight the “Bottom Solder Paste” document and click “Run”.
- Verify that the files were created in your CAM directory.
- Click the “Save” button in the “Define CAM Documents” dialog.

- Create the Aperture Report (**aperture\_report.rep**)
  - On the “File” pull down, select “CAM” (if not in the CAM mode already)
    - Click the “Aperture Report” button.
    - Browse to your CAM directory and give a name of “aperture\_report.rep”
    - Click “OK”
    - Verify that the file was created in your CAM directory.

```

Photo-Plotter Apertures Report
=====
Position Width Hgt/ID Shape Qty
=====
24 9 0 SQR 100
25 22 0 SQR 4
26 20 0 SQR 192
27 5 0 RND 226
28 4 0 RND 24
29 20 0 RND 200
30 1 0 RND 634
31 2 0 RND 4
32 19 0 SQR 50
33 32 0 SQR 2
34 30 0 SQR 96
35 10 0 RND 792

```

You have now created all of the fabrication documents that are needed to get this PCB built. Go into your CAM directory and create a ZIP file with all of the fab files in it. For this design you should have 25 files. This ZIP file is now sent to the fab shop.

NOTE: You have setup all of the CAM files for this design. If you make changes to the design, you do not need to go through the setup process again. You just go into the “Define CAM Documents” dialog, highlight all of the documents, and click “RUN”. All of the CAM files will be regenerated. It is a good practice to always regenerate all of the CAM files immediately before zipping them up and sending to fab. This ensures that they are up to date.