

2008 년 안산일대 디지털정보통신학과

***CAD* 강의용 자료**

PADS 2007

Learning the PADS User Interface

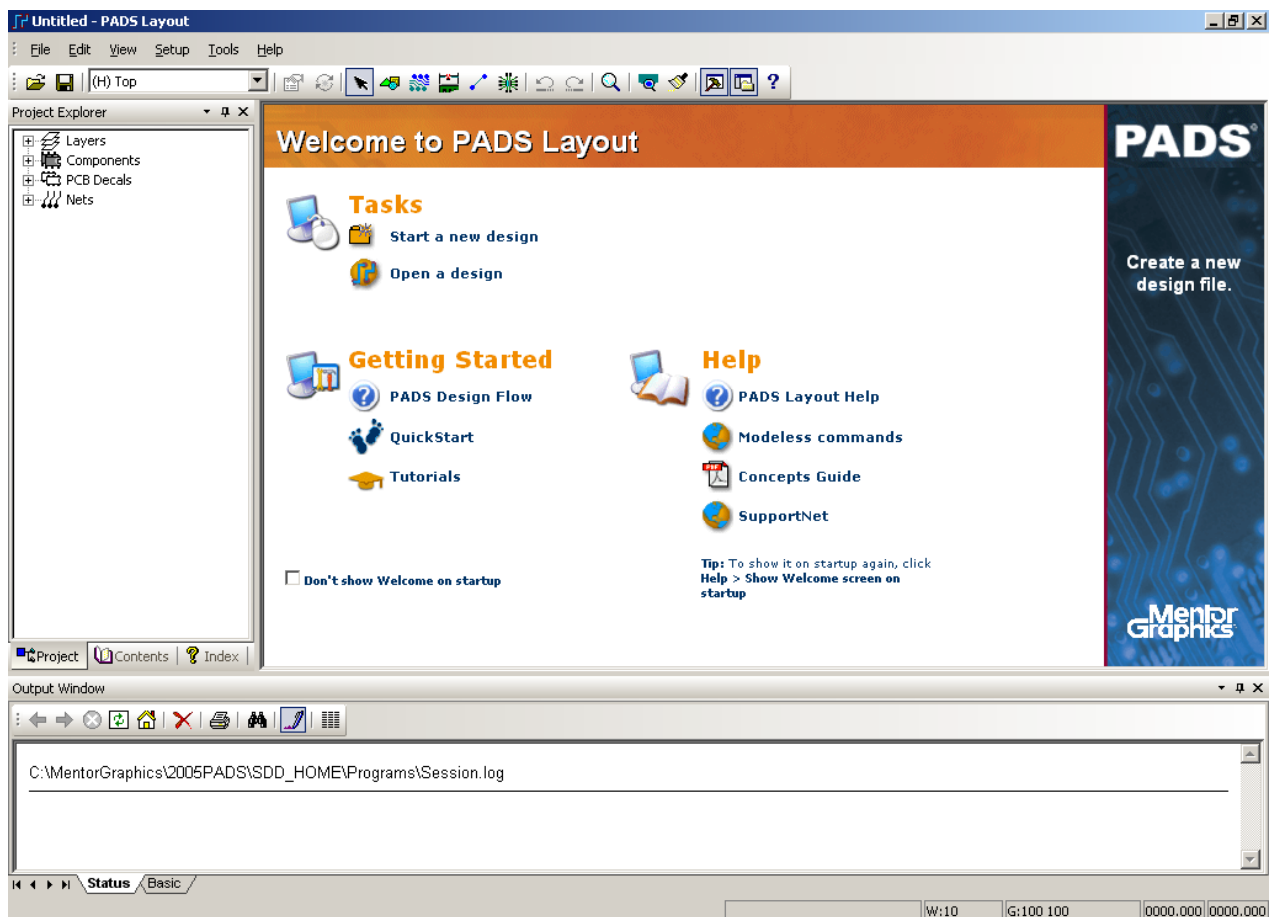
What you will learn:

- Modeless Commands
- Panning & Zooming
- Object Selection Methods

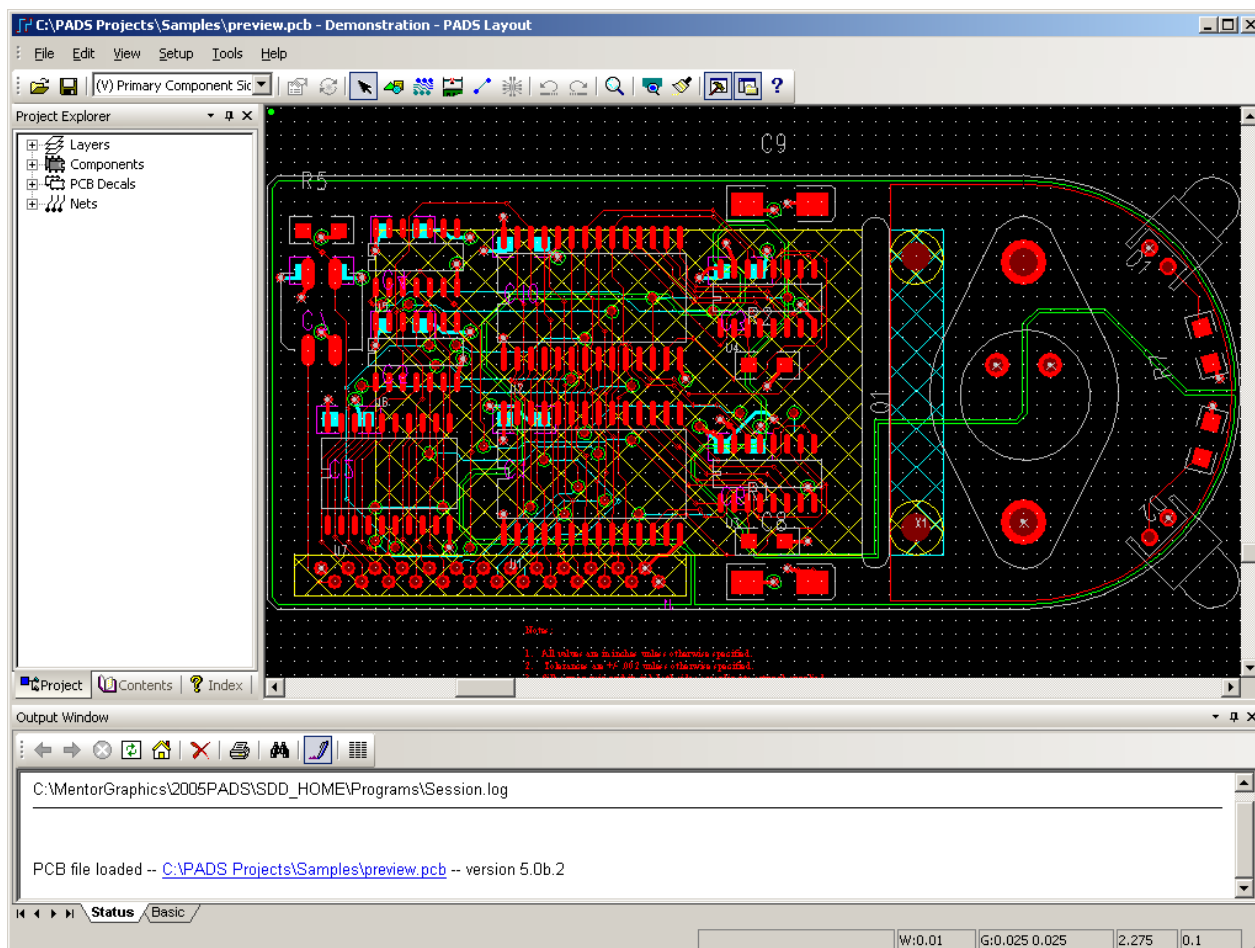
Note: This tutorial will use PADS Layout to demonstrate the user interface. Unless otherwise noted, the techniques shown below also apply to PADS Logic.

1. Begin by opening a sample PCB Design.

1.1 Start PADS Layout. Select **Start > Programs > Mentor Graphics SDD > PADS2005 SP2 > PCB Layout > PADS Layout**. The PADS Layout welcome screen should appear.



1.2 Open a sample PCB Design. Select **File > Open**. Browse to **C:\PADS Projects\Samples** and open **preview.pcb**. You should see a PCB layout similar to the one below...



1.3 You can use this sample PCB file to test the user interface features described in this tutorial.

2. Modeless Commands

Modeless commands are simple shortcut keys that can be used to change settings or perform a given action at any time. To enter a modeless command, simply type the specified shortcut key(s) followed by any necessary parameters and press enter. When you begin typing a modeless command, the Modeless Command window will appear. In addition to displaying the user's input, the Modeless Command window will also provide realtime help for the command currently being entered.

A few of the available modeless commands are listed below. For a complete list, select **Help > Modeless Commands**.

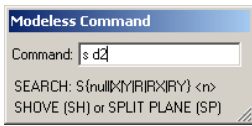
Shortcut key	Description
UN <n>	Multiple undo command. <n> is the number of operations to undo
RE <n>	Multiple redo command. <n> is the number of operations to redo.
S <s>	Part search. <s> is the reference designator.
G <n>	Set the design grid spacing to <n>.
GD <n>	Set the display grid spacing to <n>
M	Display the shortcut menu for the current mode.

Note: In PADS Layout, the grid commands use the design units specified in **Tools > Options > Global > Design Units**. Although this setting will default to mils, the

preview PCB file is set to inches.

As an example, try searching for the part labeled D2:

Enter the search shortcut key, s, followed by the part reference, d2.




This will bring component D2 into view and center the mouse pointer over it.

3. Panning & Zooming

There are several methods available to move about a design and adjust the magnification level.

3.1 Using Zoom Mode.

You can enter zoom mode by selecting **View > Zoom**, pressing **Ctrl + W**, or by clicking the  button on the toolbar.

While in zoom mode, the following features are available:

- Zoom in one level – Left Click.
- Zoom out one level – Right Click.
- Zoom in by a specific amount – Click and hold the left mouse button in the center of the area you wish to view and drag the mouse up. Release the left mouse button when the desired zoom level is shown.
- Zoom out by a specific amount – Click and hold the left mouse button in the center of the area you wish to view and drag the mouse down. Release the left mouse button when the desired zoom level is shown.

When you are finished zooming, you can exit zoom mode by pressing escape.

3.2 Panning

You can pan around the design by using any of the following methods:

- Scrollbars – You can use the scrollbars at the right edge and bottom of the design window to pan vertically and horizontally.
- Mouse Scroll Wheel – If your mouse is equipped with a scroll wheel, you can use it to pan vertically.
- Insert Key – To center the design view on the mouse pointer, press the insert key.

3.3 View Sheet / Board

To view the entire sheet (PADS Logic) or board (PADS Layout) press the Home key.

3.4 Middle Mouse Button

If your mouse has a scroll wheel or middle mouse button, the following features will be available:

- Center View – Click and release the middle mouse button to center the view on the mouse pointer.

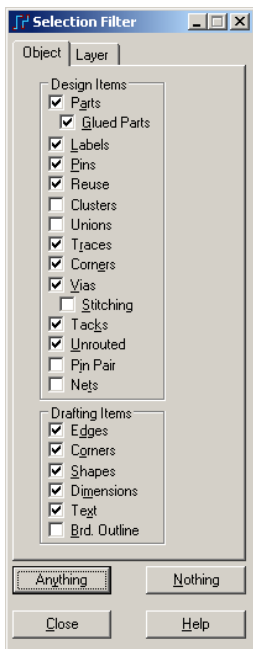
- Zoom In – Click and hold the middle mouse button in the center of the area you wish to view and drag the mouse up. Release the middle mouse button when the desired zoom level is shown.
- Zoom Out – Click and hold the middle mouse button in the center of the area you wish to view and drag the mouse down. Release the middle mouse button when the desired zoom level is shown.

4. Object Selection Methods

4.1 Selection Filter

The selection filter will allow you to 'filter out' the object types that you are not interested in, making it easier to select specific components.

To access the selection filter, select **Edit > Filter...** or press **Ctrl+Alt+F**. The selection filter window for PADS Layout is shown below.

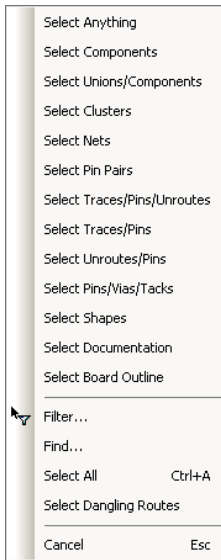


Checked objects are able to be selected, unchecked objects are filtered out. In PADS Layout it is also possible to filter out specific layers of the pcb by using the Layer tab.

As an example, open the selection filter and press the Nothing button. Then select the Vias check box and close the selection filter window. You will now only be able to select vias. If you press **Ctrl+A**, all of the vias will be selected. This could be useful if, for instance, you wanted to quickly change the size or type of all of the vias. Press escape or click in an empty area to unselect the vias.

4.2 Selection Filter Shortcuts

There is a faster method of changing the filter options. With nothing in the design view selected, right click to display the shortcut menu. The top portion of this menu consists of selection filter presets and is shown below.



Selecting a filter preset is the same as making the corresponding changes in the selection filter window. For example, selecting the **Select Components** preset is the same as only checking Parts and Glued Parts in the filter window.

Return the selection filter to it's default state by right clicking and selecting **Select Anything** from the top of the menu.

4.3 Selection Cycle (PADS Layout Only)

In crowded pcb designs it may become difficult to select a particular object, even with the use of the selection filter. To overcome this problem you can cycle through all the nearby objects by using the Tab key.

To demonstrate this feature, select the top left pin of part U3. (Hint: Use the modeless search command, S, to locate U3 quickly.) Now repeatedly press the tab key to cycle between nearby parts and traces.

This completes the user interface tutorial.

Creating a new Part Type using the PCB & CAE Decal Wizards

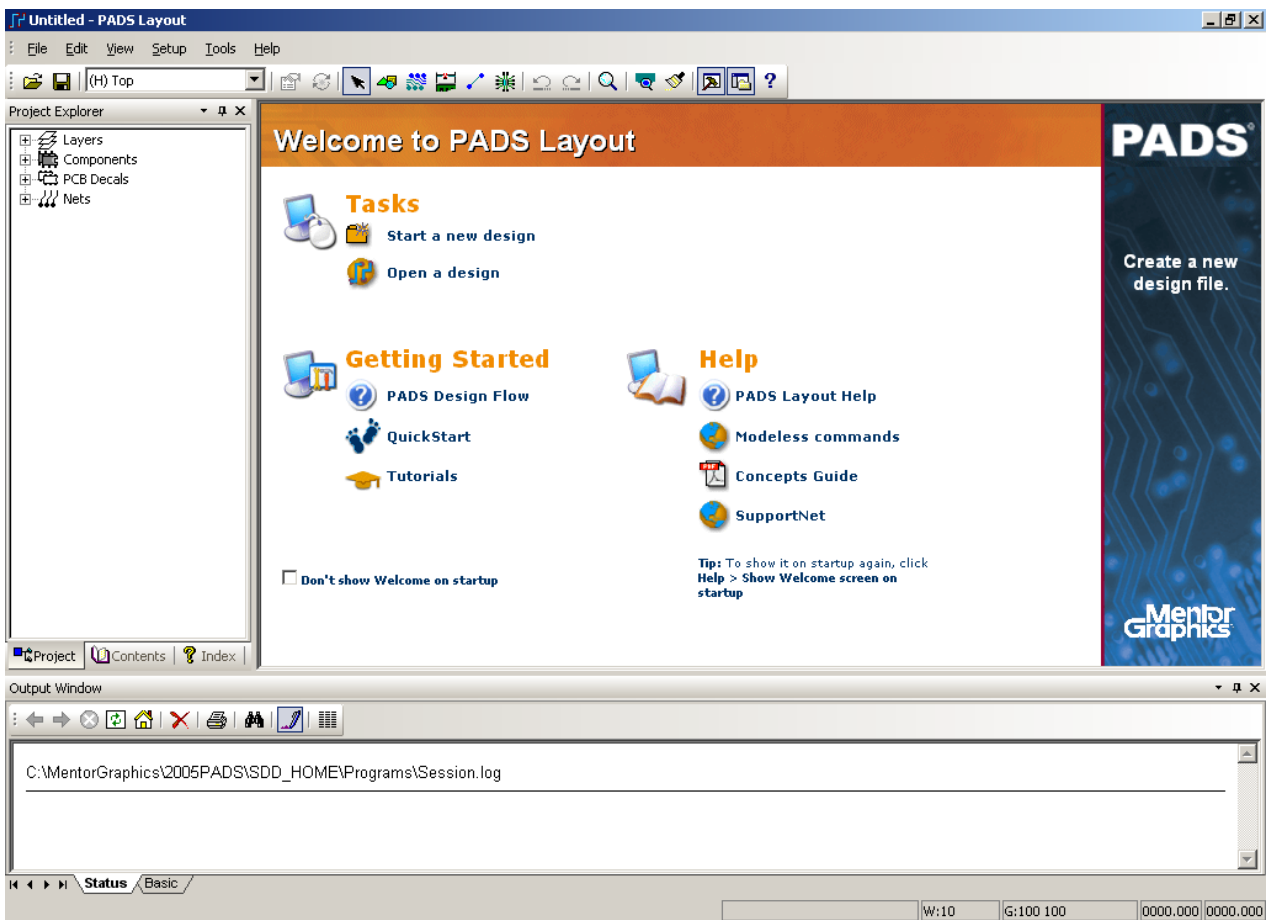
What you will learn:

- Creating and Managing User Libraries
- Creating a PCB Decal
- Creating a CAE Decal
- Creating a Complete Part

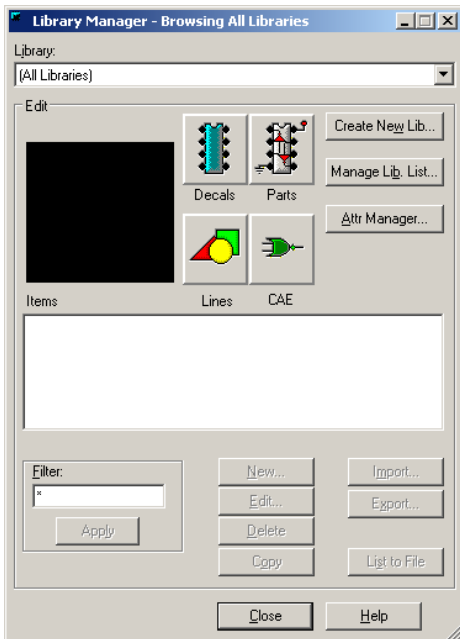
1. Create a custom user library

Before creating your first new part type, you will need to setup a user library. This library will be saved to your network shared drive, allowing you to access it from any computer in the VCU School of Engineering.

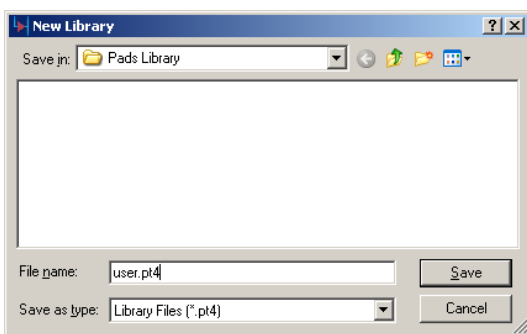
1.1 Start PADS Layout. Select **Start > Programs > Mentor Graphics SDD > PADS2005 SP2 > PCB Layout > PADS Layout**. The PADS Layout welcome screen should appear.



1.2 Open the Library Manager. Select **File > Library...**



1.3 Press the **Create New Lib...** button. Browse to the **H: Drive** and create a new folder named **Pads Library**. Open this folder and type **user.pt4** in the file name field. Then press the Save button.



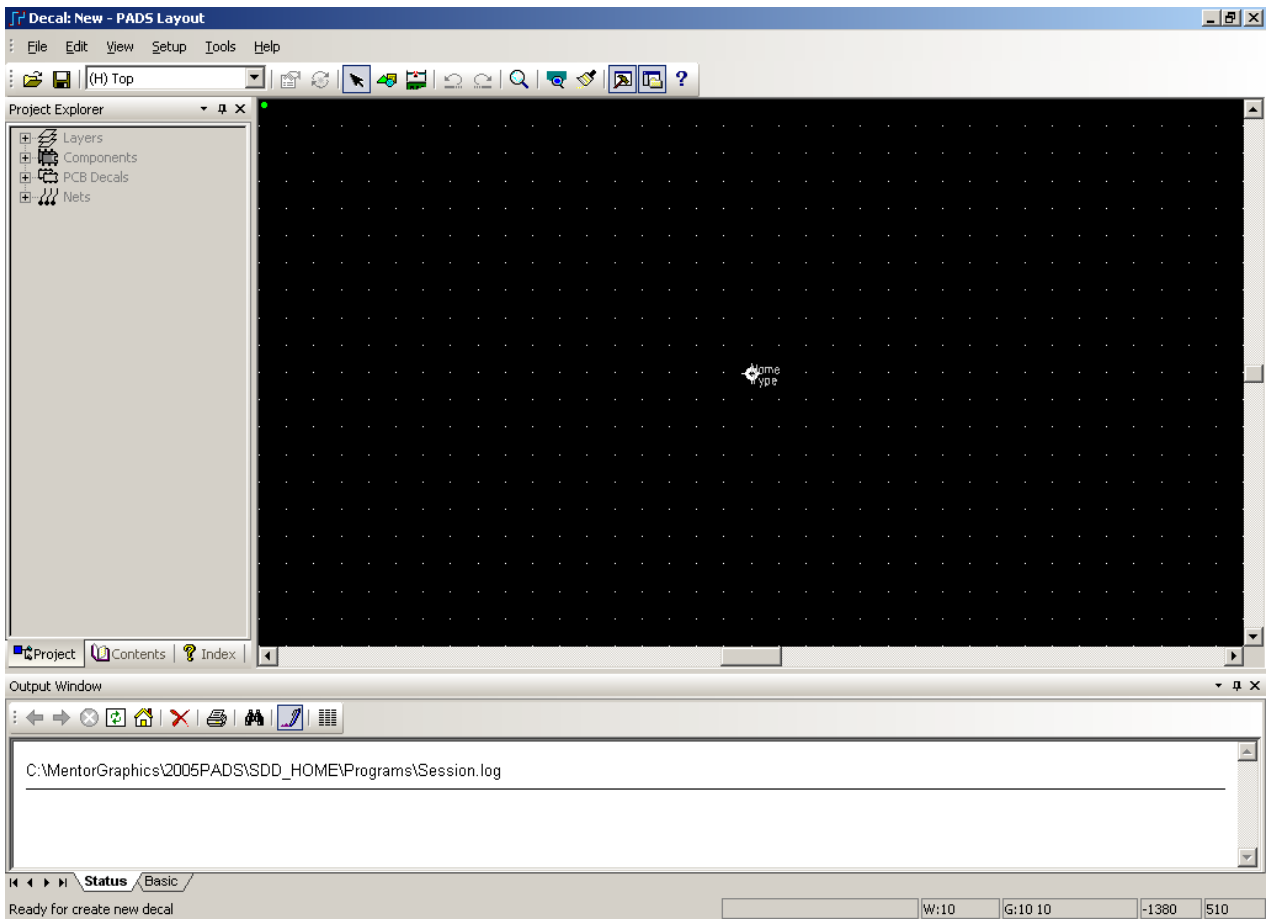
1.4 Once you have created the new library, close the library manager.


Note: To access this library in future windows sessions: open the library manager and press the **Manage Lib List...** button. Then press **Add...**, browse to **H:\Pads Library** and select the **user.pt4** file.

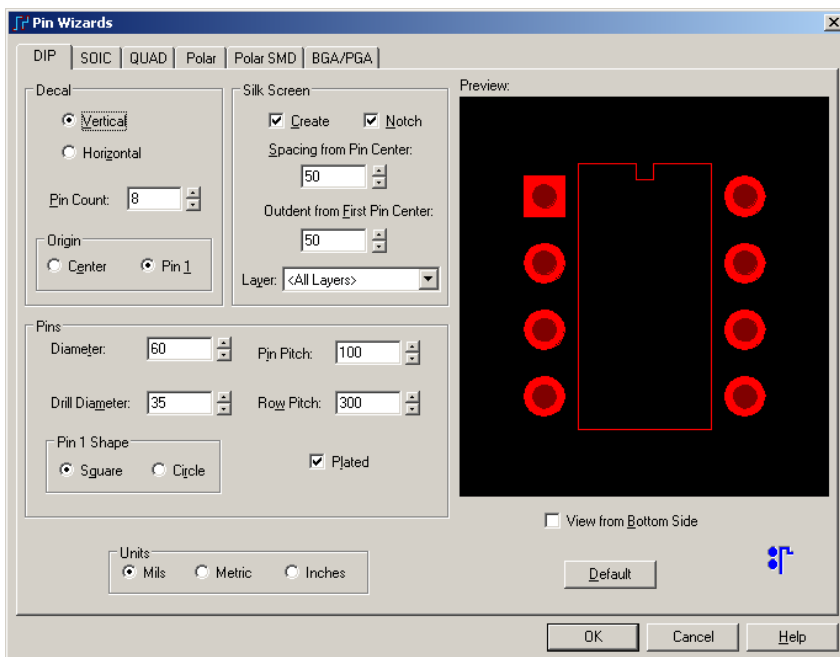
2. Create a PCB footprint using the PCB Decal Wizard

We will now create a PCB footprint for the National Semiconductor LM2675M-3.3 voltage regulator chip. The relevant datasheet can be found [here](#). (Page 24 contains information on the part's physical dimensions).

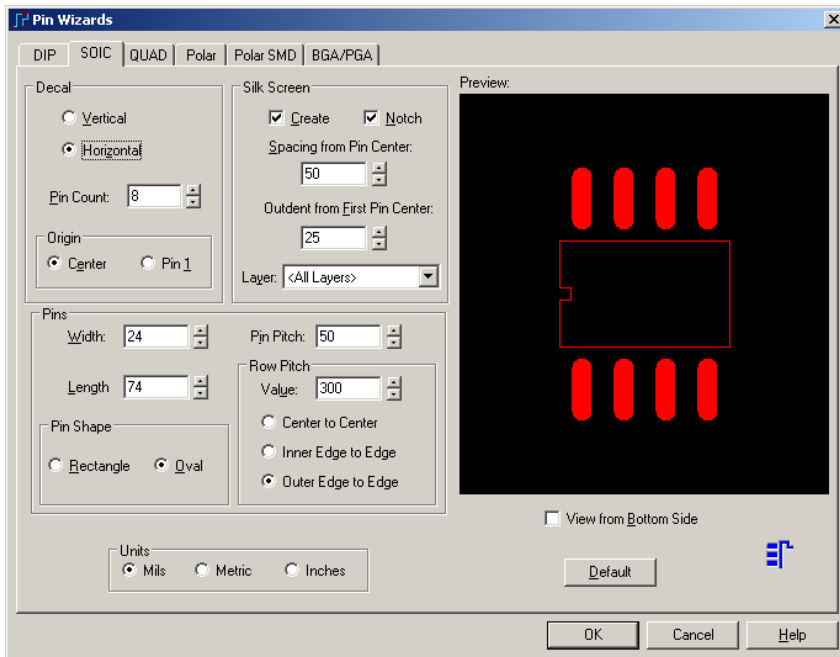
2.1 Open the PCB Decal Editor. In PADS Layout, select **Tools > PCB Decal Editor**. The PADS Layout window should now look similar to the image below.



2.2 Start the PCB Decal Wizard. Enable the Drafting Toolbar by pressing the button. From the Drafting Toolbar press the Wizard button: . The Pin Wizards window will now appear.



2.3 We will be creating a decal for the 8-pin Small Outline Package, so select the SOIC tab.



2.4 Make the following changes:

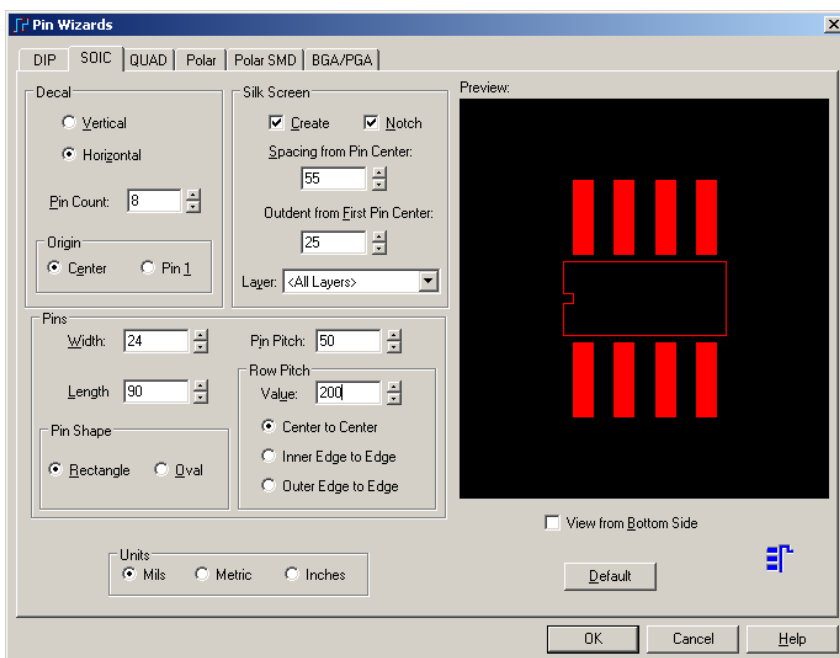
In the Decal area, select Horizontal and type 8 in the Pin Count box.

In the Silk Screen area, type 55 in the Spacing from Pin Center box.

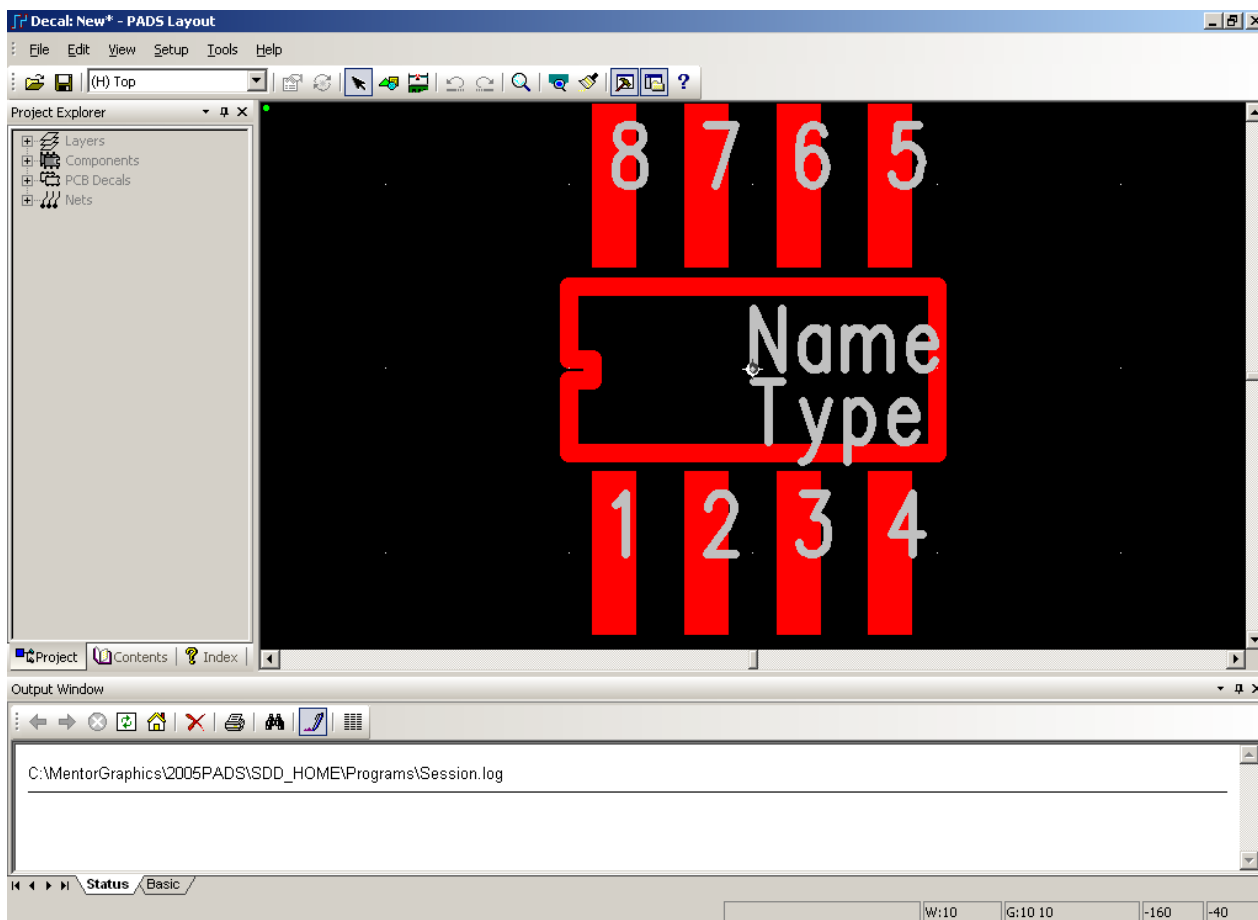
In the Pins area, type 24 in the Width box, 90 in the Length box and 50 in the Pin Pitch box.

In the Pin Shape area, choose Rectangle.

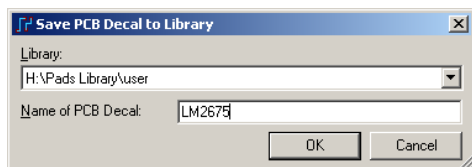
In the Row Pitch area, type 200 in the Value box, and select Center to Center.



2.5 When the Pin Wizards window looks like the one above, press OK. You will now return to the PCB Decal Editor which should look like the image below.



2.6 Save the PCB footprint to your user library. Select **File > Save Decal**. In the Save PCB Decal to Library dialog, select **H:\Pads Library\user** for the library and type **LM2675** in the Name of PCB Decal field.



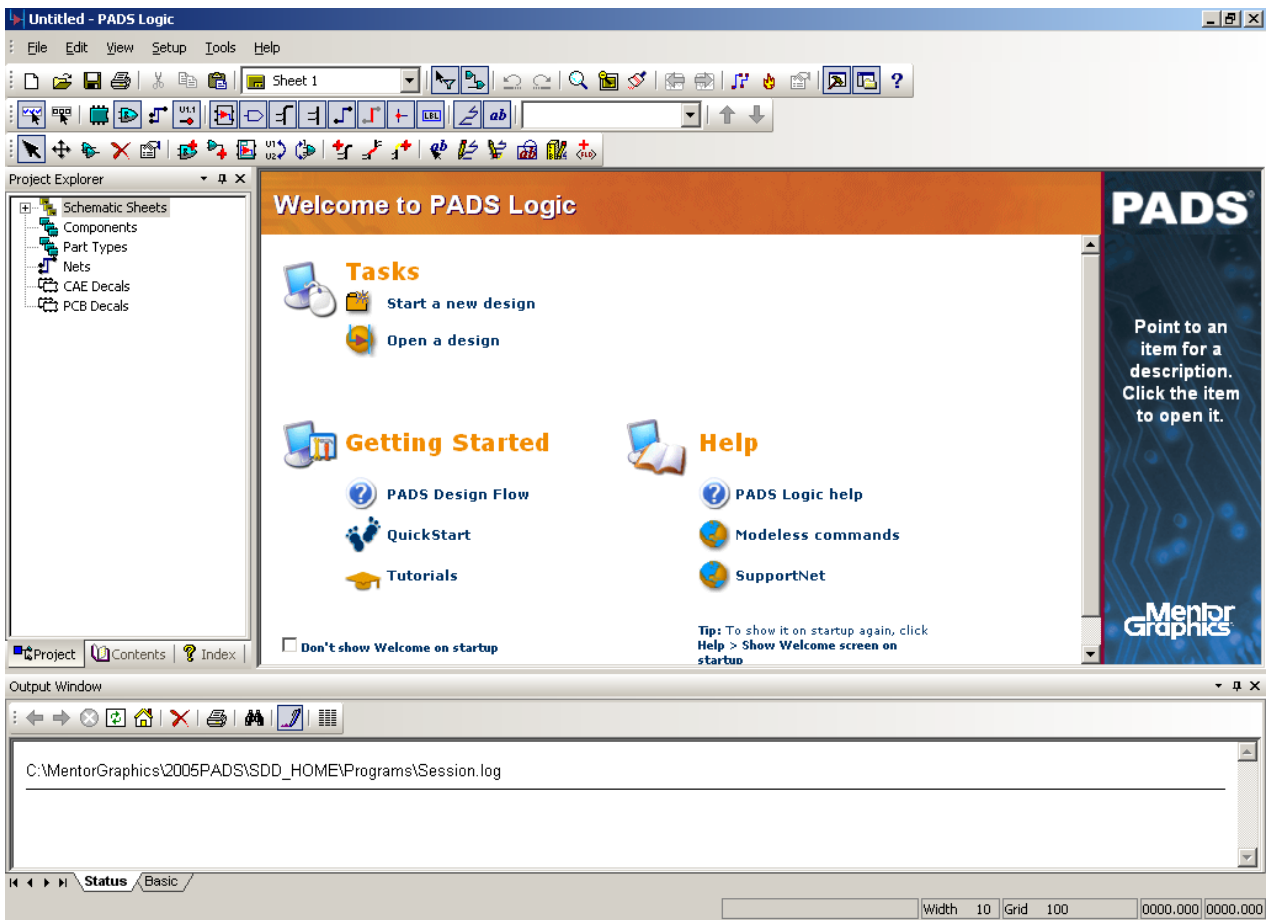
2.7 Press OK. When it asks if you want to create a new Part Type, say yes, then press OK in the Save Part Type to Library dialog box.

2.8 Exit the PCB Decal Editor and PADS Layout. Select **File > Exit Decal Editor** to return to PADS Layout. Then select **File > Exit** to close PADS Layout.

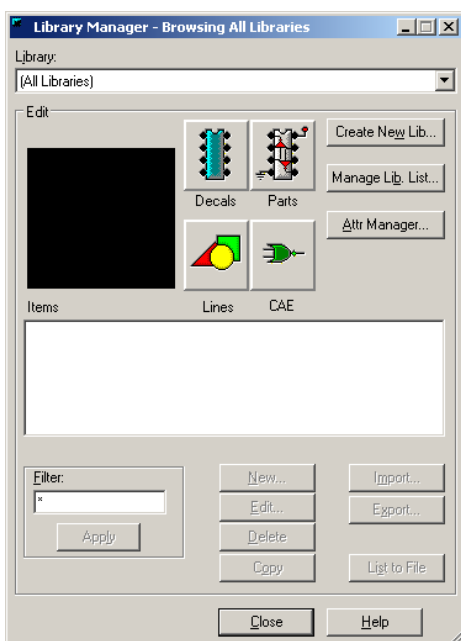
3. Create a schematic symbol using the Part Editor & CAE Decal Wizard.

We will now create a schematic symbol for the National Semiconductor LM2675M-3.3 voltage regulator chip.

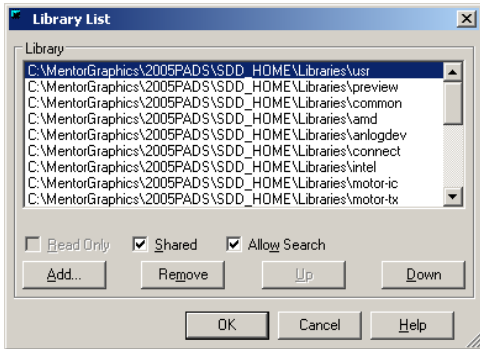
3.1 Open PADS Logic. Select **Start > Programs > Mentor Graphics SDD > PADS2005 SP2 > System Design > PADS Logic**. The PADS Logic welcome screen should appear.



3.2 Add your user library to the Library List. Select **File > Library...**

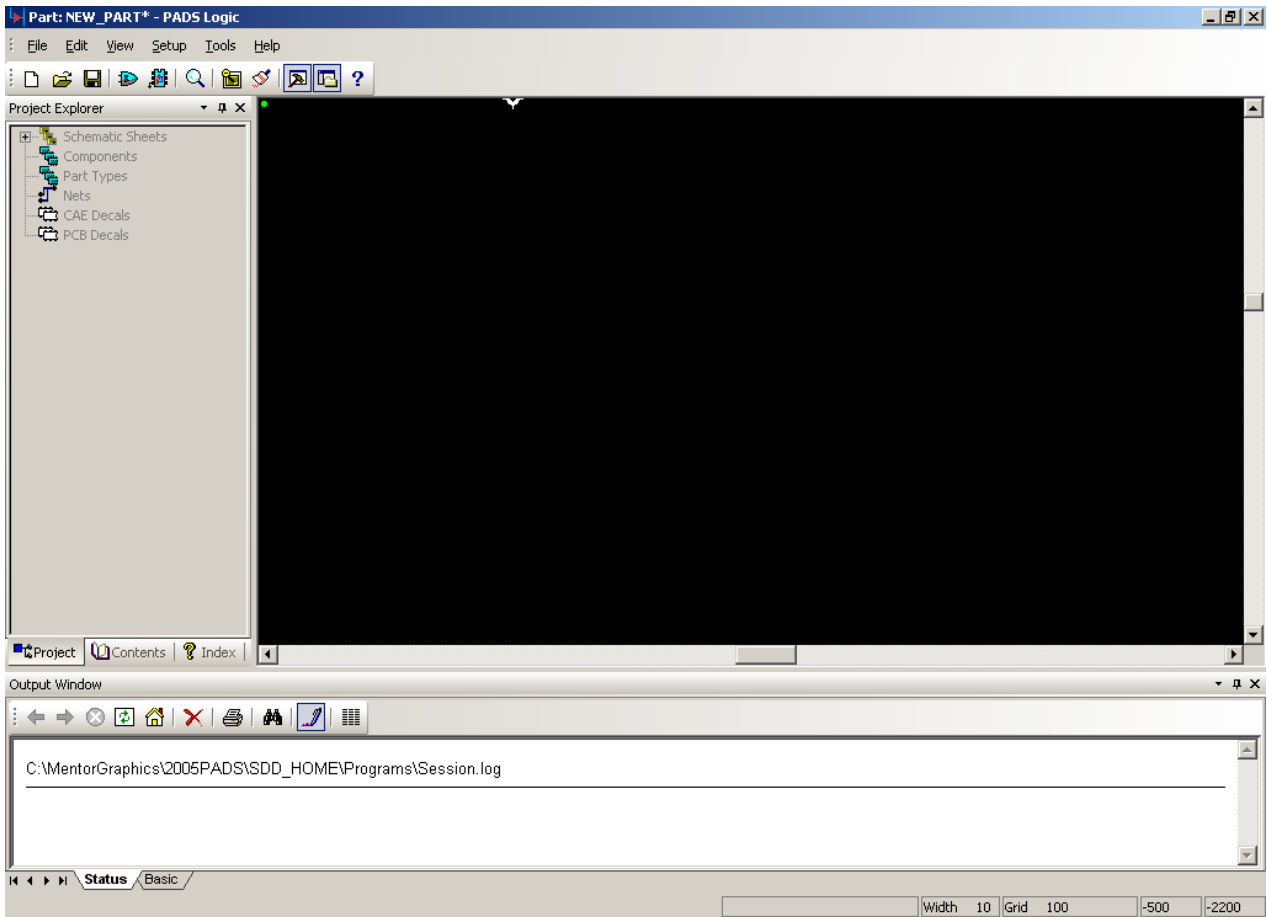


Then press the **Manage Lib List...** button.



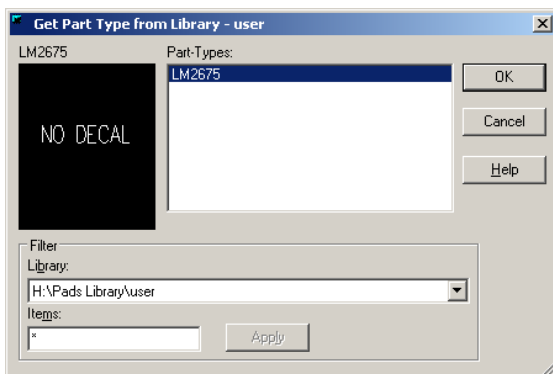
Press **Add...**, browse to **H:\Pads Library** and select **user.pt4**.
After adding your user library, press OK and then close the library manager.

3.3 Open the Part Editor. Select **Tools > Part Editor**. The PADS Logic window should now look similar to the image below.

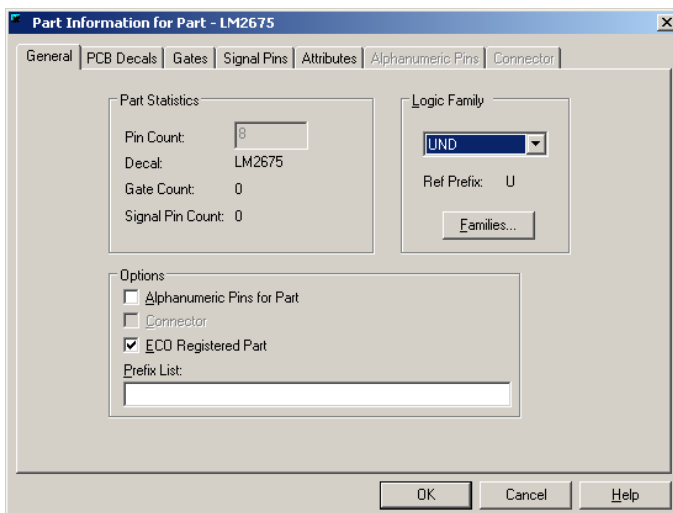


3.4 Open the LM2675 part type that you created in part 2 of this tutorial. Select **File > Open...** On the Select type of editing item dialog, choose Part Type and press OK.

The Get Part Type from Library dialog will appear. In the filter area, select the **H:\Pads Library\user** library and type ***** in the items box. Press Apply. Now select LM2675 from the Part-Types list and press OK.



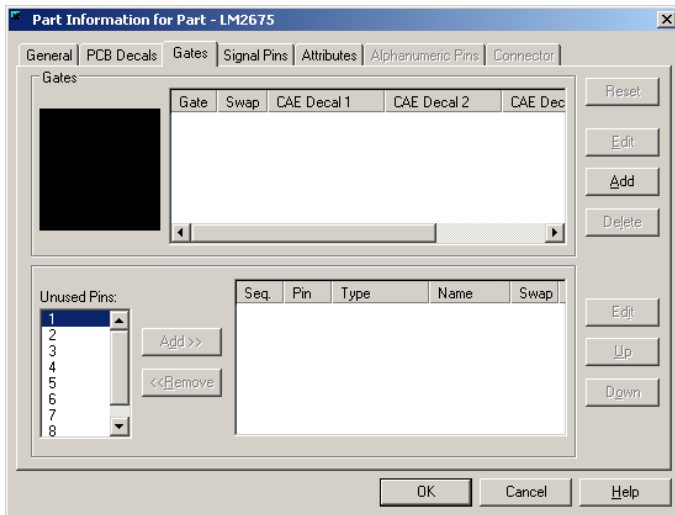
3.5 Edit the part's electrical properties. Press the  button on the toolbar to open the Part Information window.



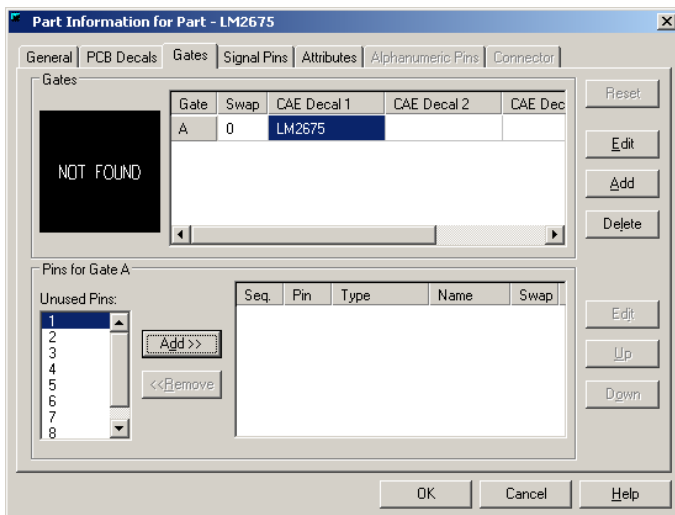
Under Logic Family, select ANA.

3.6 Specify the (not yet created) schematic decal.

Select the Gates tab.



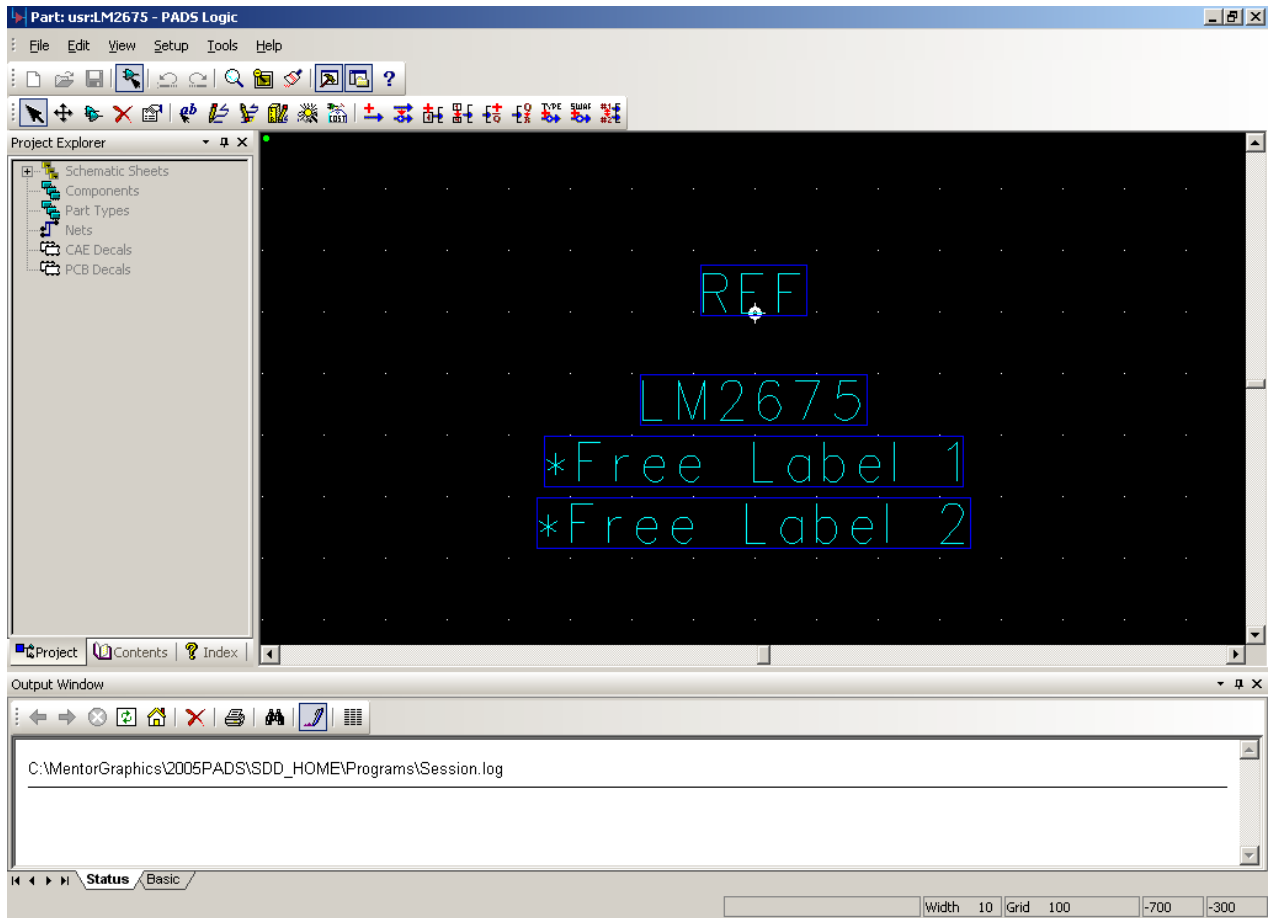
Press the Add button. Click in the CAE Decal 1 field, type LM2675 and press Enter. Then press the OK button.




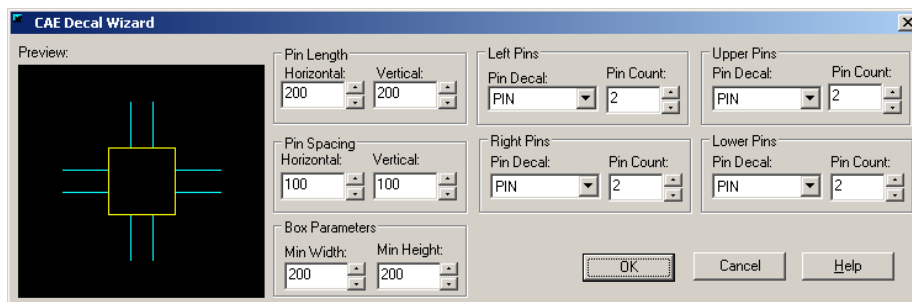
3.7 Open the CAE Decal Editor. Press the  button on the toolbar to open the CAE Decal Editor. The Select Gate Decal dialog will pop up.

Press OK to continue. PADS Logic will display a warning stating that the selected decal doesn't exist and will be created. Press OK.

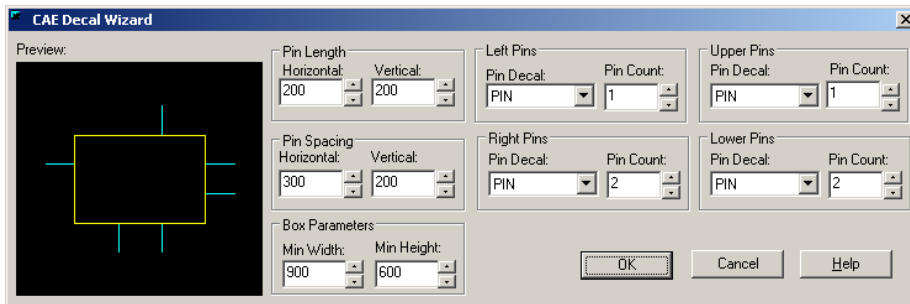
The CAE Decal Editor should now appear.



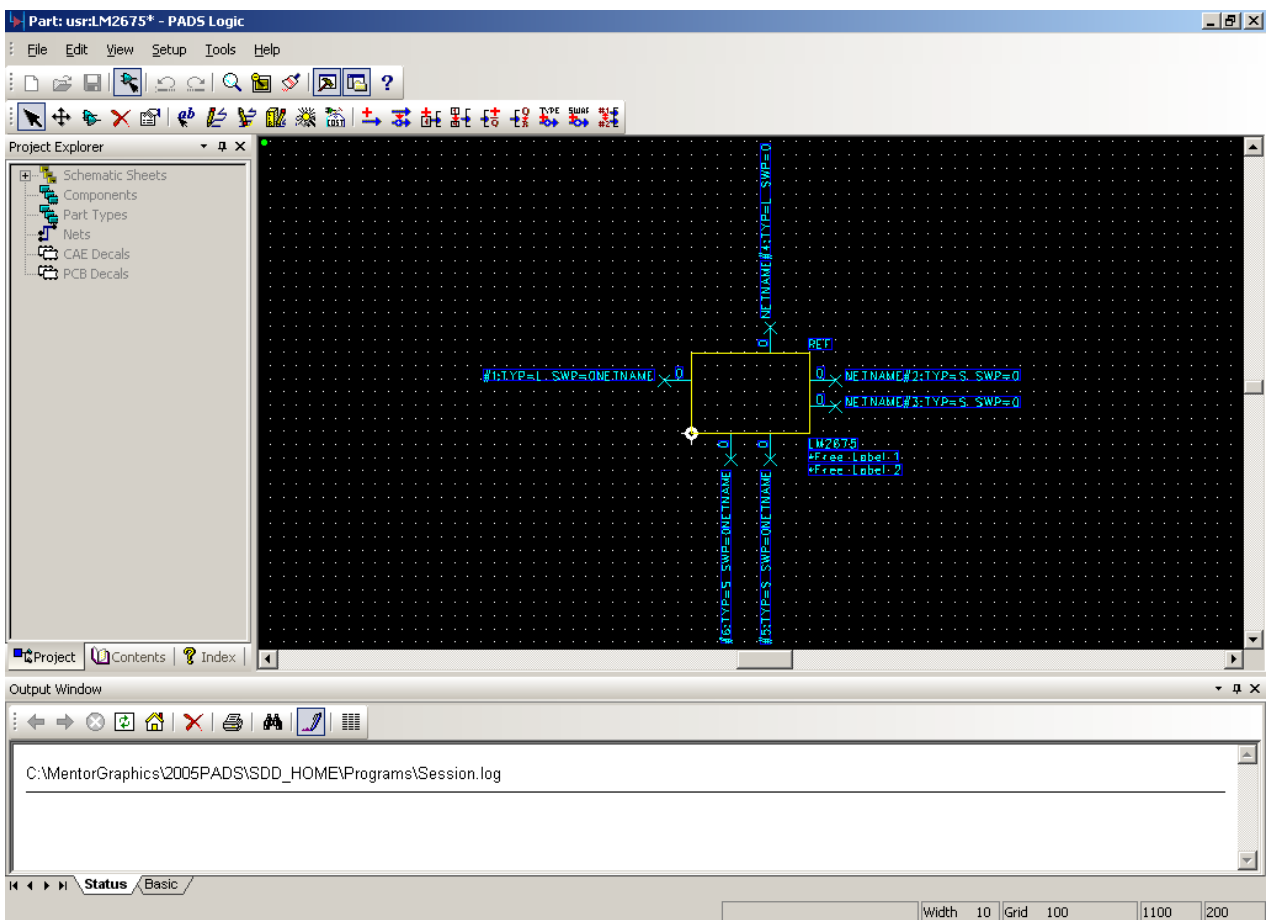
3.8 Start the CAE Decal Wizard. Press the  button on the Decal Editing toolbar to launch the Decal Wizard.



3.9 Make the following changes:
In the Pin Spacing area, type 300 in the Horizontal box and 200 in the Vertical box.
In the Left Pins area and Upper Pins area, set the Pin Count to 1.
In the Right Pins area and Lower Pins area, set the Pin Count to 2.

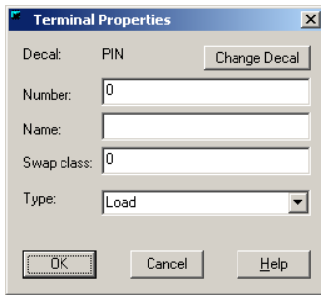


When the specified values have been entered, press OK. The CAE Decal window should now look similar to the one below.



Note: You may wish to zoom in on the decal while making changes to the pins. To enter zoom mode, press **Ctrl + W**. Use the left mouse button to zoom in and the right mouse button to zoom out. Press Esc to exit zoom mode.

3.10 Name and Number the pins. Right-click in an empty area and select the **Select Terminals** menu item. Double click on the leftmost pin. The Terminal properties dialog will open.



Type 7 in the number box and type VIN in the Name box. The process for the remaining 5 pins. Proceeding clockwise, naming follows:

Number: 4, Name: FB

Number: 1, Name: CB

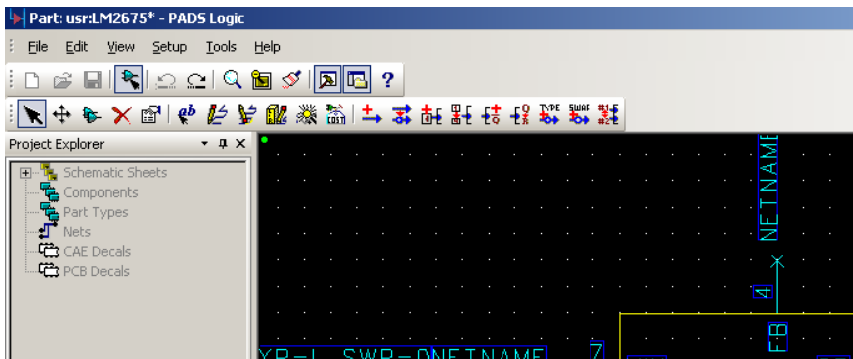
Number: 8, Name: VSW

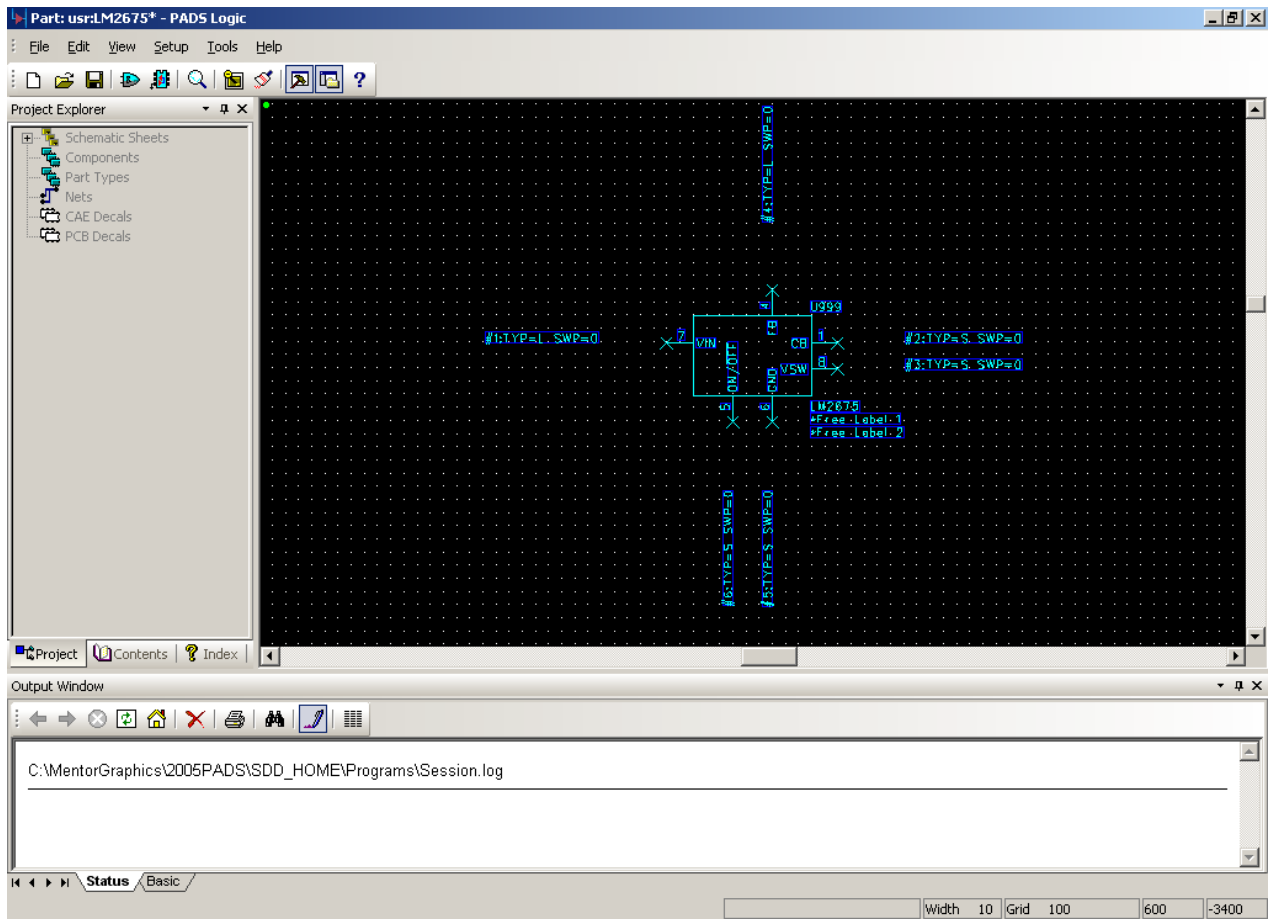
Number: 6, Name: GND

Number: 5, Name: ON/WOFF

Note: For pin 5, the name will appear as ON/OFF. The backslash is not and will make a bar appear over the word immediately following.

When you are finished naming and numbering the pins, the Component will be similar to the image below.





Select **File > Save**. Then Select **File > Exit Part Editor**.

You can now use the LM2675 part type in your PADS Logic and PADS Layout designs.

Creating a simple 3.3V Regulator Circuit using PADS

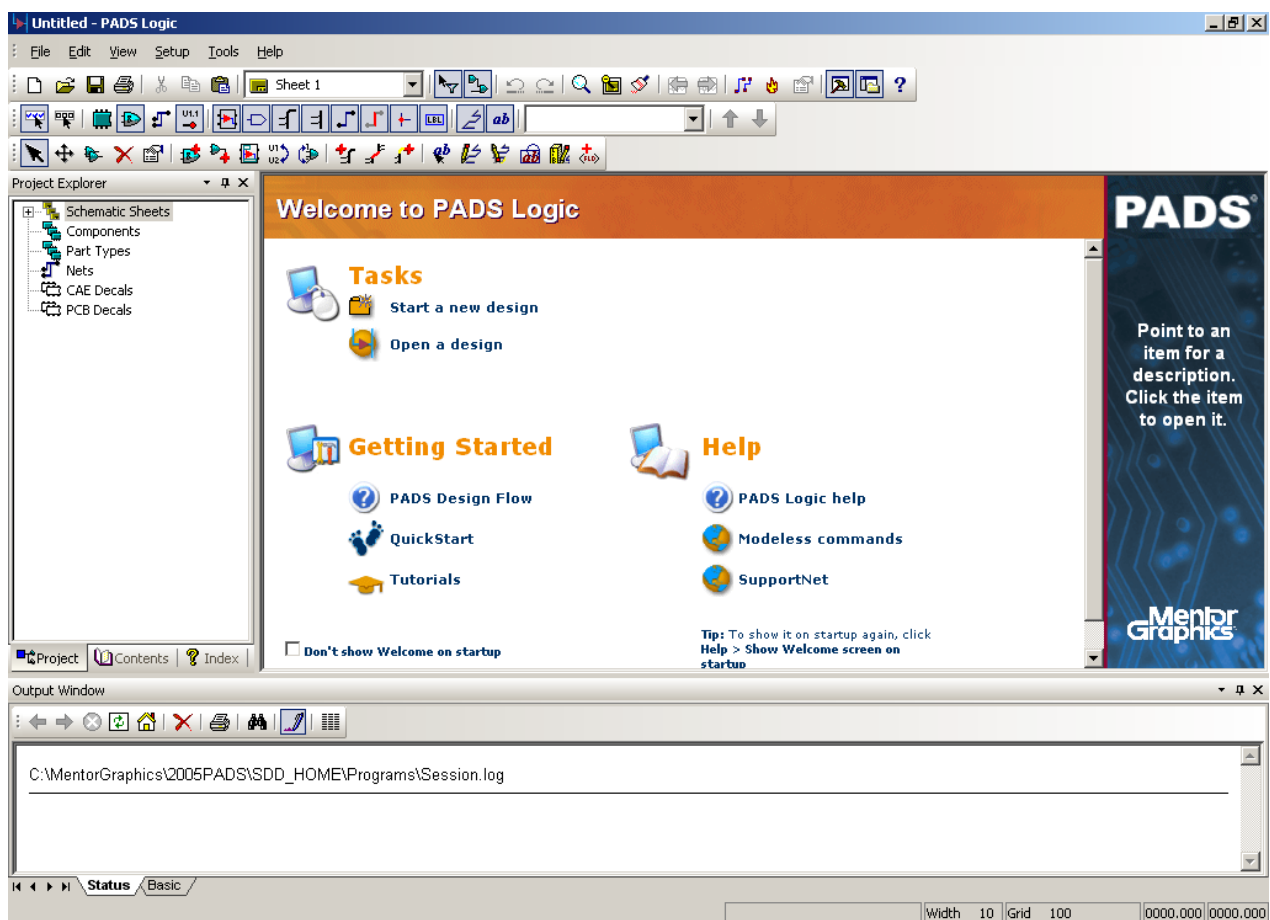
What you will learn:

- Creating a Circuit Schematic with PADS Logic
- Creating a PCB Design with PADS Layout
- Routing a PCB Design with PADS Router
- Linking PADS Logic to PADS Layout & PADS Router
- Generating Gerber Files with PADS Layout

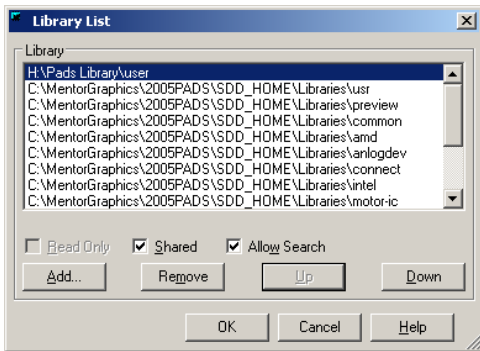
Note: This tutorial assumes that you've completed the PADS User Interface tutorial and the New Part Type tutorial.

1. Creating a Circuit Schematic using PADS Logic

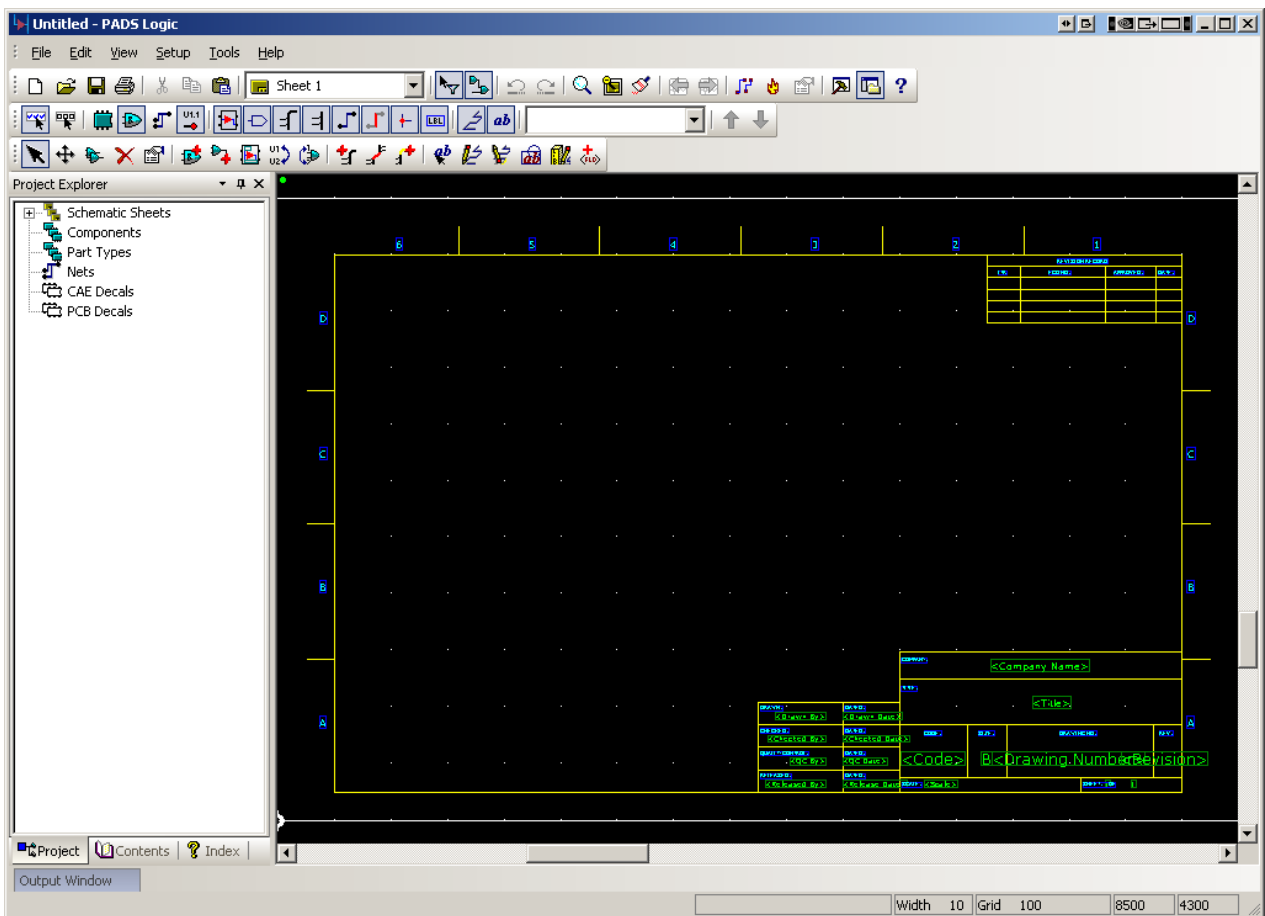
1.1 Begin by opening PADS Logic. Select **Start > Programs > Mentor Graphics SDD > PADS2005 SP2 > System Design > PADS Logic**.




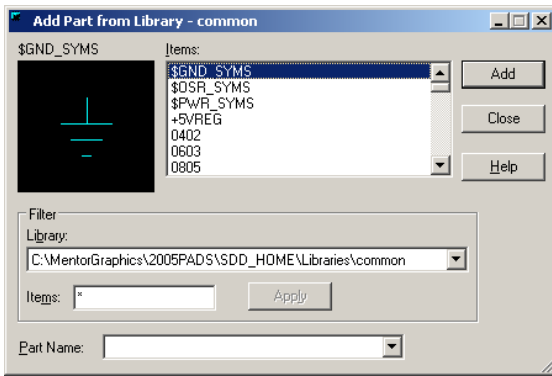
1.2 Ensure that your user library is loaded by using the library manager – **File > Library**. (Refer to section 3 of the New Part Tutorial for step-by-step instructions).



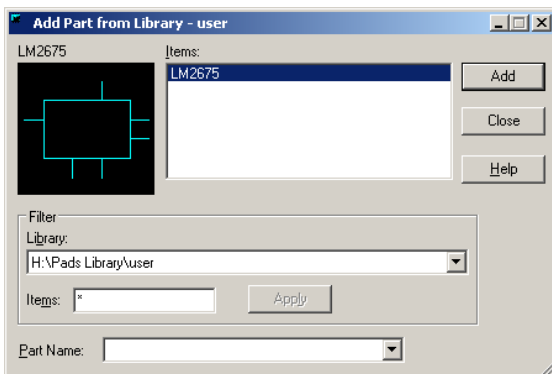
1.3 Start a new design by selecting **File > New**. Your PADS Logic window should look something like this.



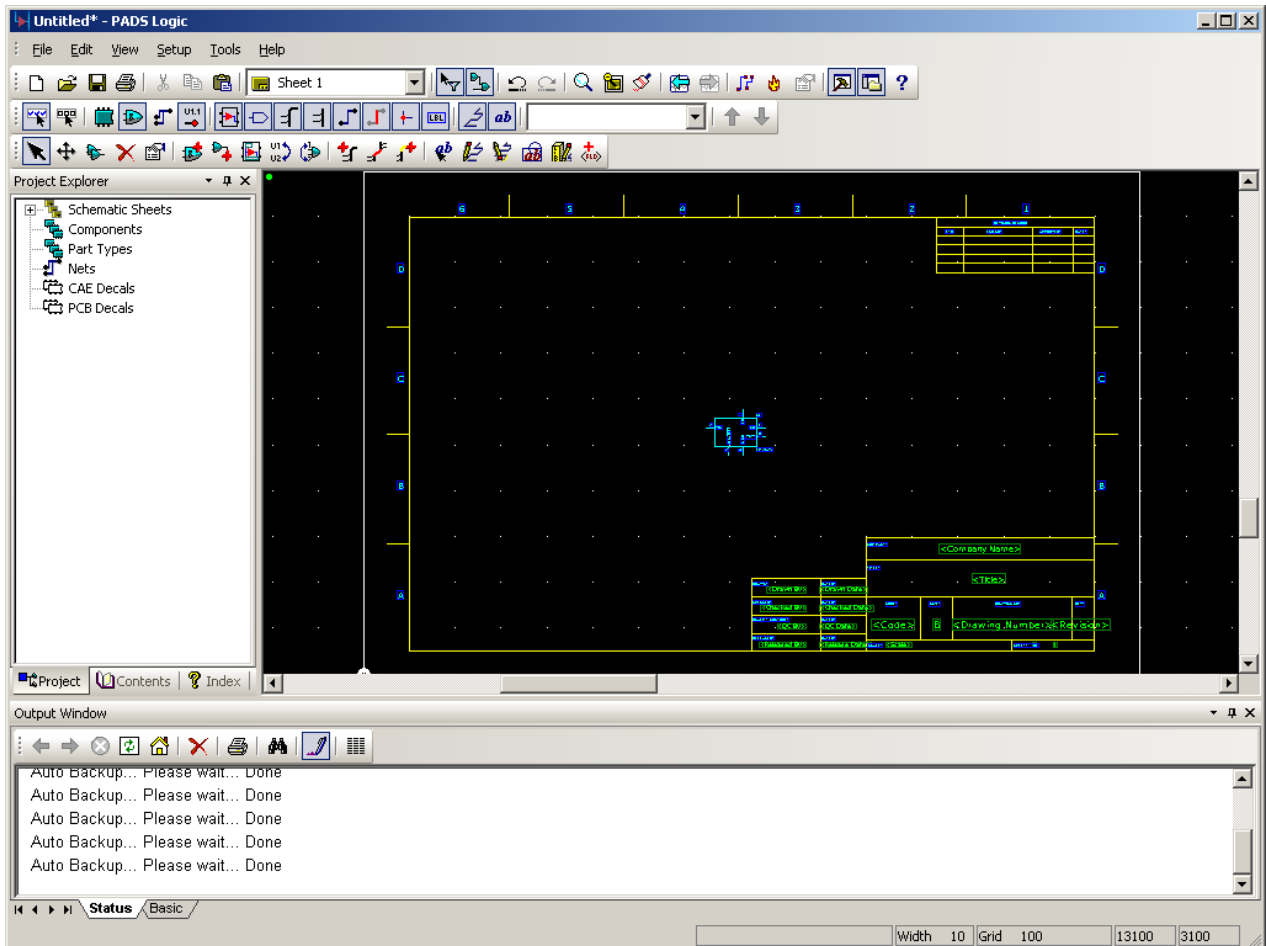
1.4 Use the Add Part window to add an LM2675 voltage regulator to your design. To add a part to your design, click on the  button on the design toolbar. This will open the Add Part from Library window as shown below.



To locate the LM2675 part, you will need to select your custom library from the Library dropdown list in the filter. Select **H:\Pads Library\user**. Ensure that the Items field contains a single asterisk ***** and then press Apply. This will filter the results, displaying only the parts in your user library. The window should look like the one below.




Press the Add button. An outline of the LM2675 part will appear below the mouse pointer while it is within the design window. Close the Add Part window and click somewhere near the center of the sheet to add an instance of the LM2675 part. Press escape to exit add part mode. Your schematic window should look similar to the one below.

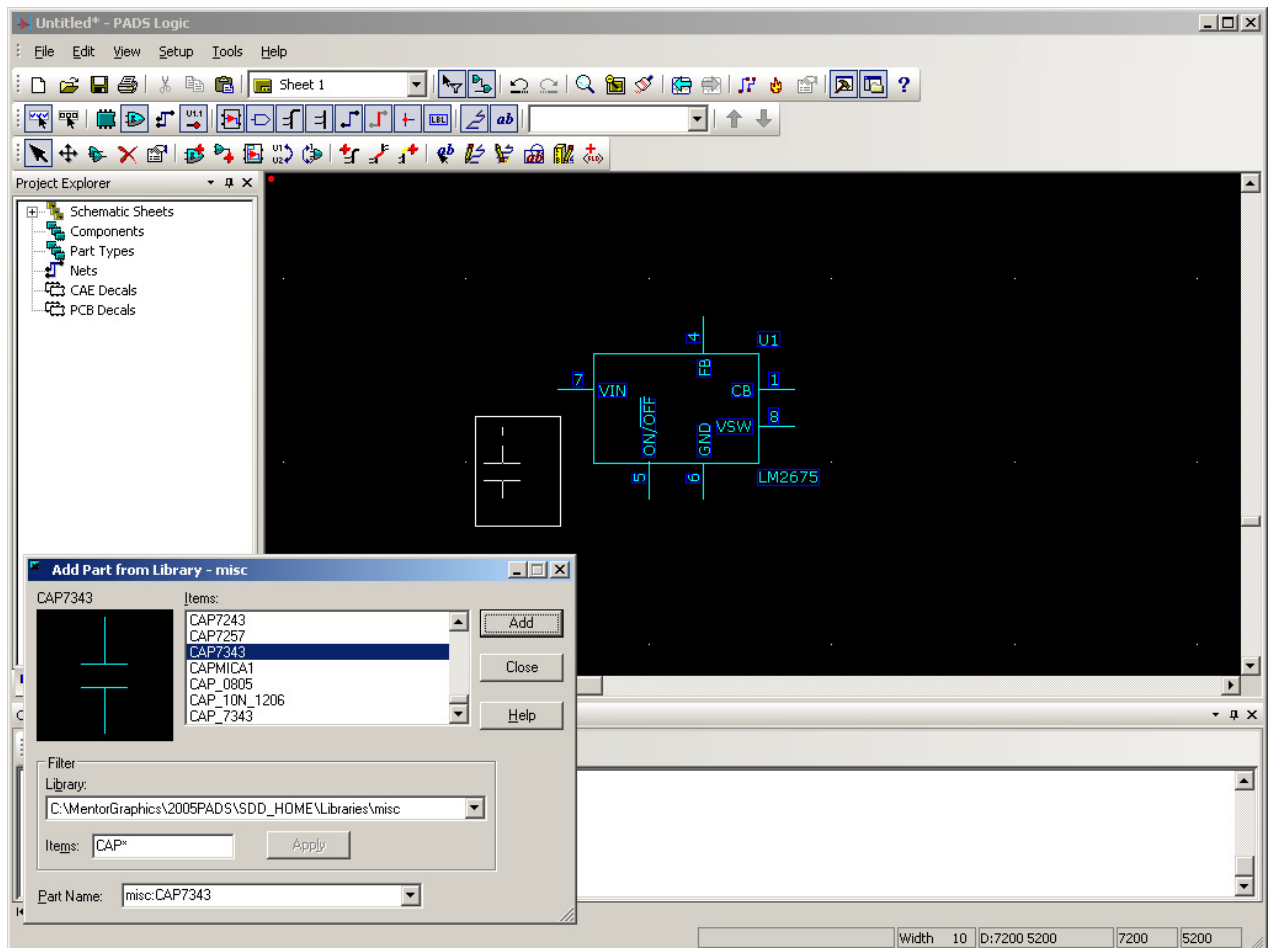


Note: You may wish to zoom in on the LM2675 before completing the following steps. (Refer to the User Interface Tutorial if necessary).

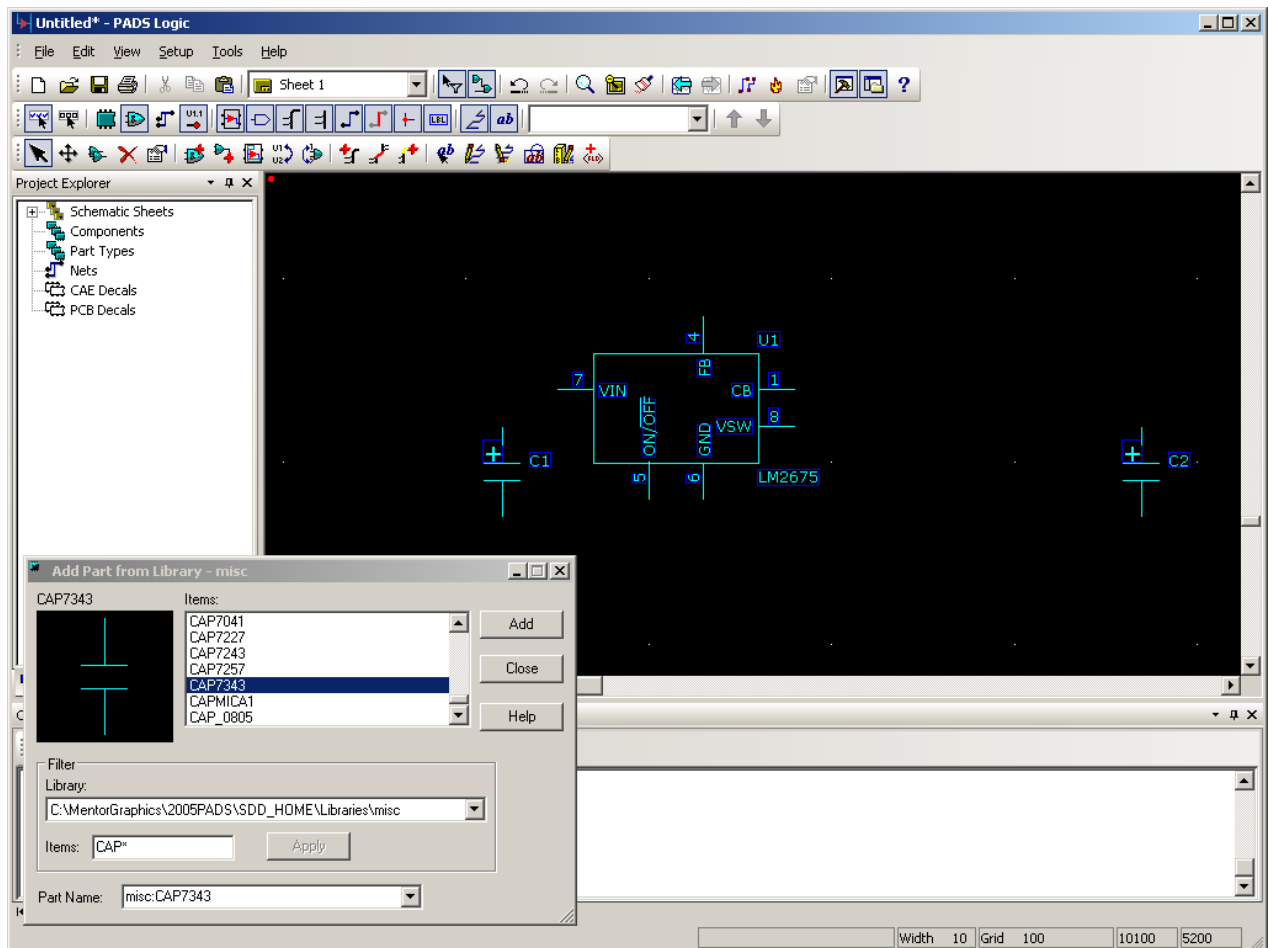
1.5 Add capacitors to your design.

Open the add part window by clicking the  button on the toolbar. In the library dropdown list, choose the **misc** library (C:\W MentorGraphics\W2005\PADS\WSD\HOME\WLibraries\Wmisc). In the Items menu, type **CAP*** and then press Apply. This will filter the results to display only the parts starting with 'CAP' that exist in the 'misc' library.

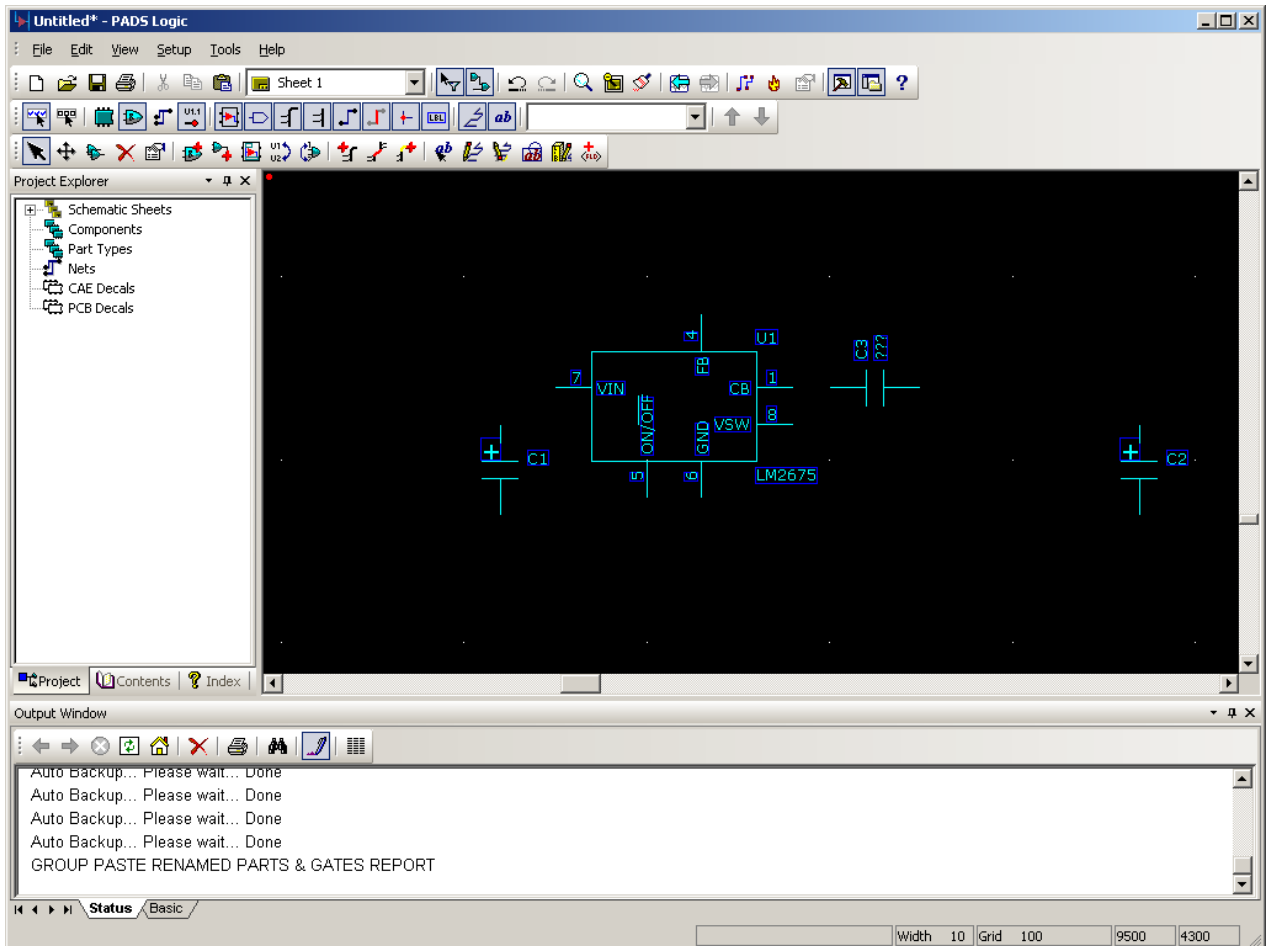
Select the part named CAP7343 from the list and press Add. Move the Add Part window so that most or all of the design area is visible. Your screen should look similar to the one below.



Place the CAP7343 capacitor to the left of the LM2675 as shown above. Place a second CAP7343 to the far right as shown in the schematic below.

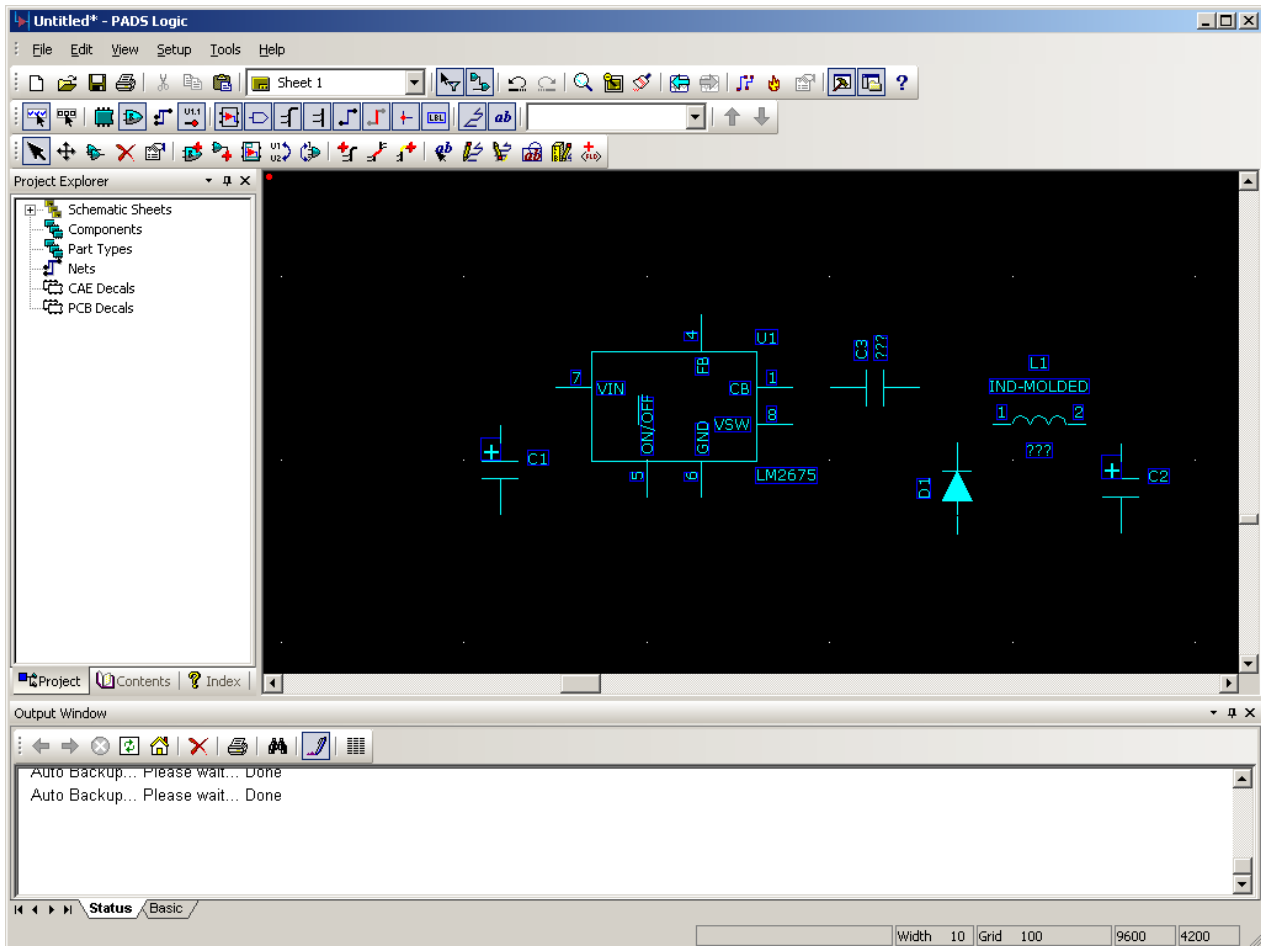


Now select CAP1206 from the Add Part window, press Add and close the window. Before adding the capacitor to your design, press **Ctrl + R** to rotate it 90 degrees. Place the capacitor to the right of the LM2675 as shown below.



1.6 Add additional circuit components to your design.

Add an inductor and diode to your design as shown in the image below. Use the IND-MOLDED part for the inductor and the DIODE part for the diode. Both of these parts are found in the 'misc' library.

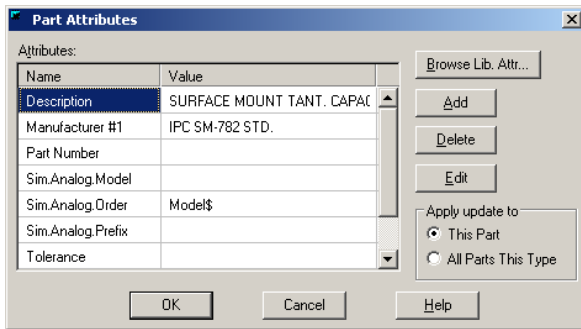


1.7 Assign values to your components.

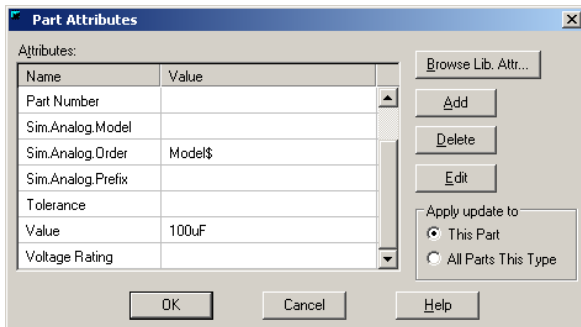
Ensure that part selection is enabled by right clicking in an empty area and selecting the **Select Gates** option. This can also be achieved by pressing the button on the filter toolbar.

Double click on the left-most capacitor. This will open the Part Properties window as shown below.

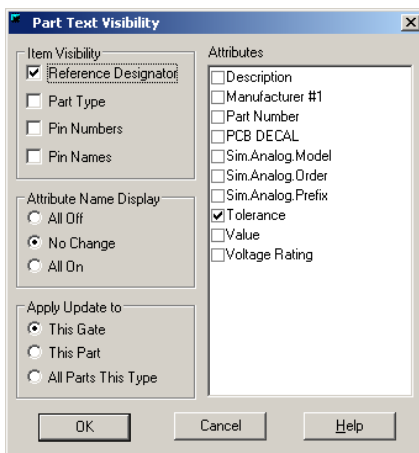
Click on the Attributes button to open the Part Attributes window.



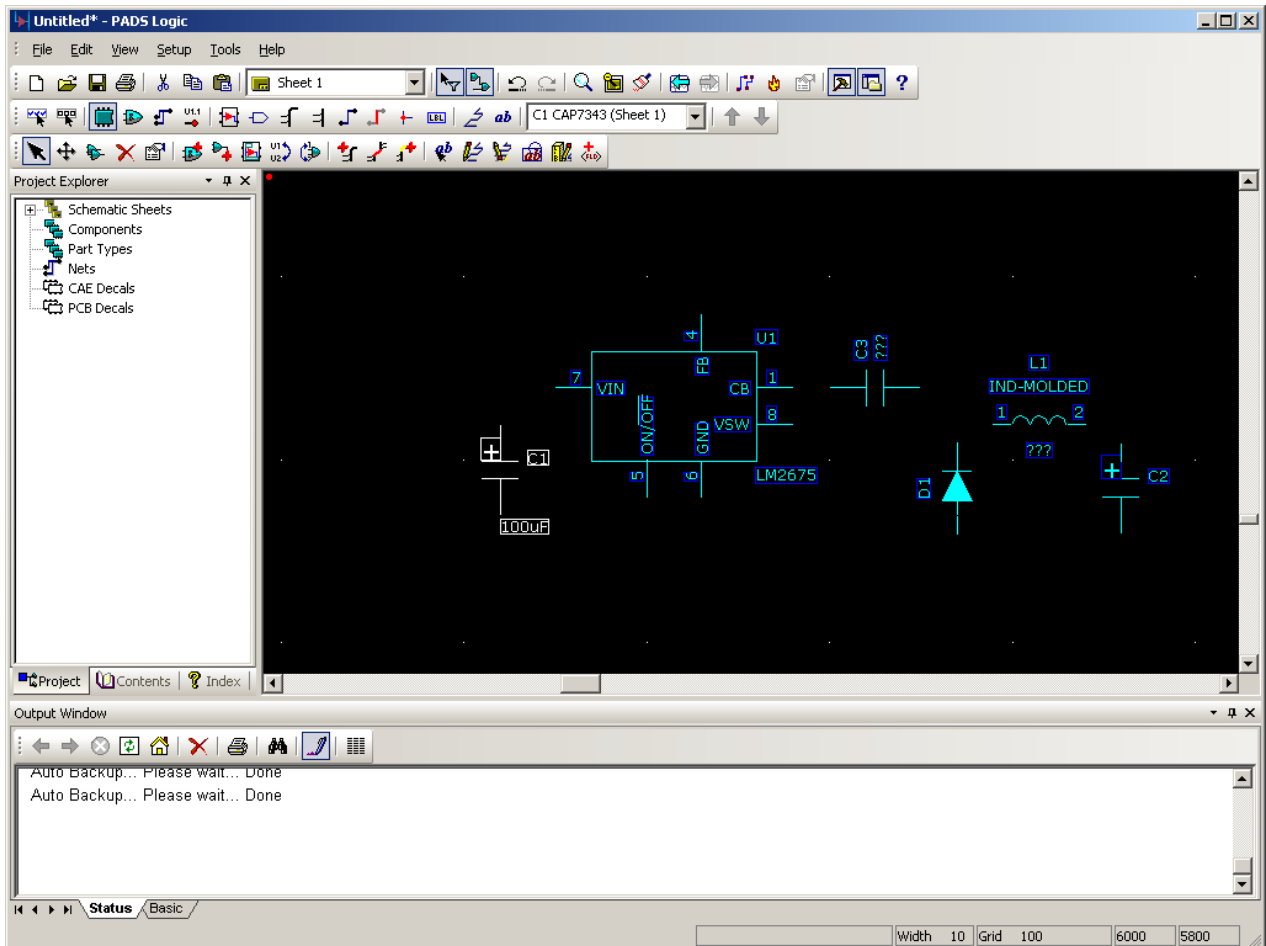
Scroll down to the Value field, double click and enter **100uF** in the text box.



Press OK to close the Part Attributes window and return to the Part Properties window. By default, the CAP7343 part will not display its value in the schematic. To change this, click the Visibility button to open the Part Text Visibility window.

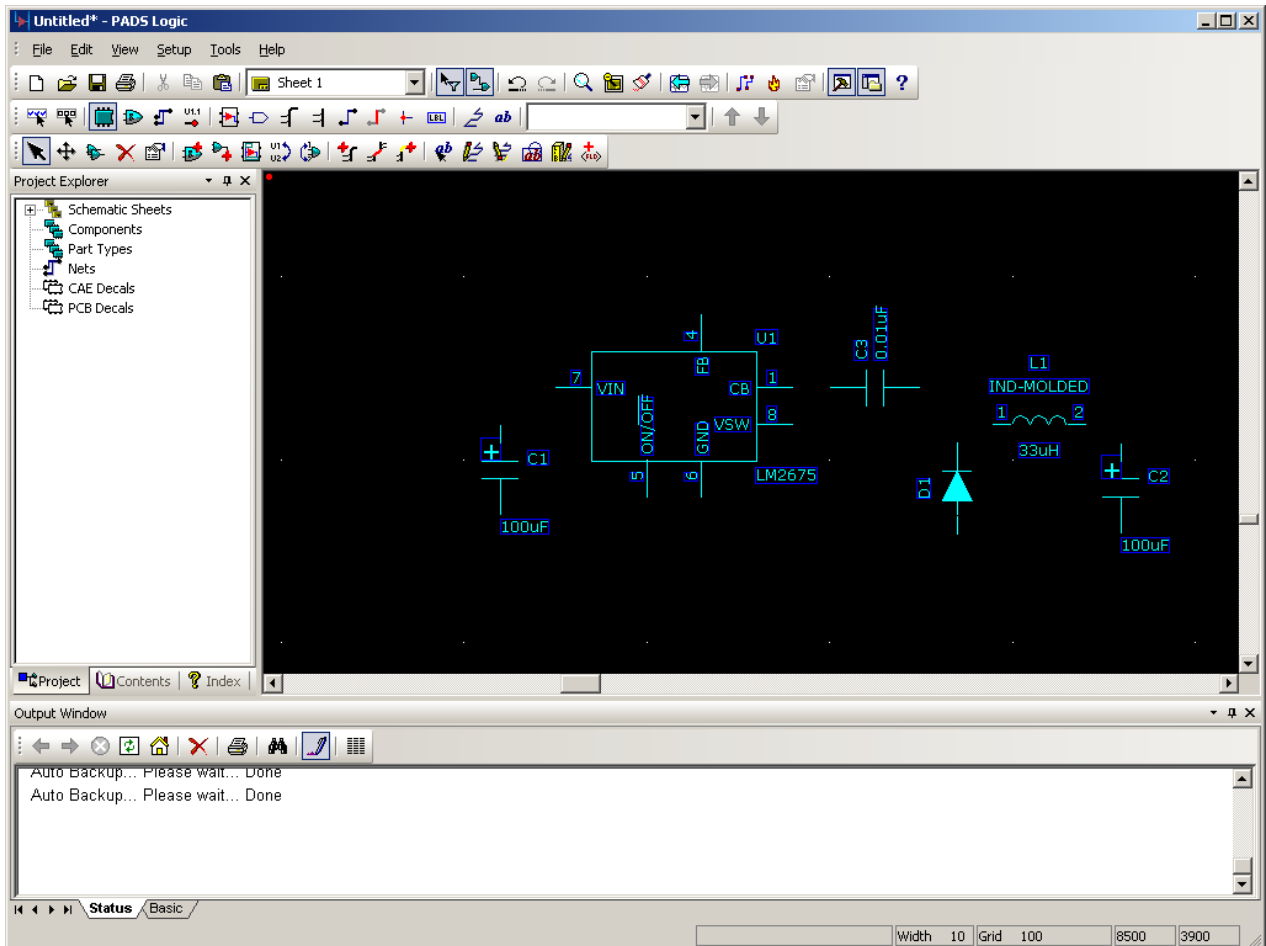


Check the Value checkbox on the right side to enable display of the capacitor's value. Press OK to close the Part Text Visibility window and then press Close to close the Part Properties window. Your schematic should now look like the one below.



Repeat this process for the other components, assigning the values listed below:

- Right Capacitor – 100uF
- Middle Capacitor – 0.01uF
- Inductor – 33uH

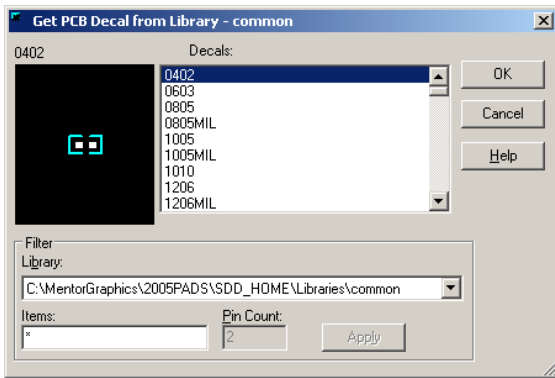


1.8 Change the PCB footprints assigned to the inductor and diode.

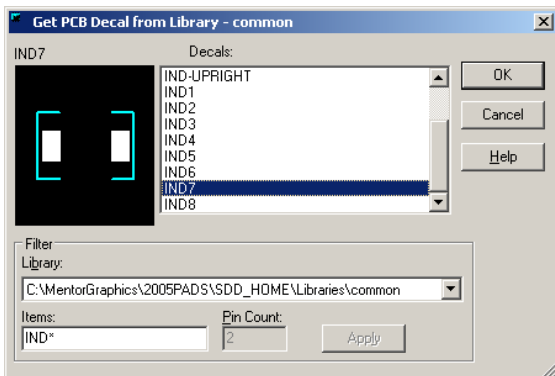
The IND-MOLDED and DIODE parts both default to using through-hole footprints. We will change these parts to use surface mount footprints.

Double click the inductor to open the Part Properties window. Click on the PCB Decals button to open the PCB Decals Assignment window.

Click Browse... The Get PCB Decal from Library window will appear. This window functions almost identically to the Add Part window.

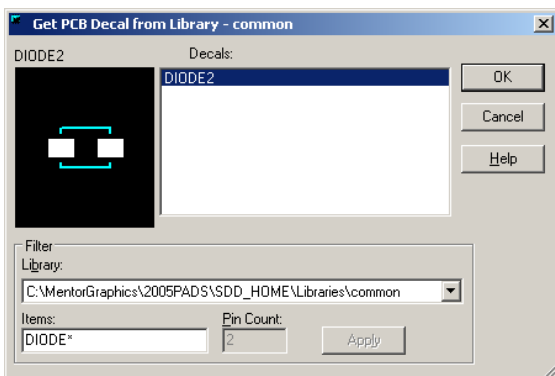


The footprint we will be using for the inductor is named IND7 and is found in the 'common' library.



Select this decal and press OK. Then press OK in the PCB Decal Assignment window and close the Part Properties window.

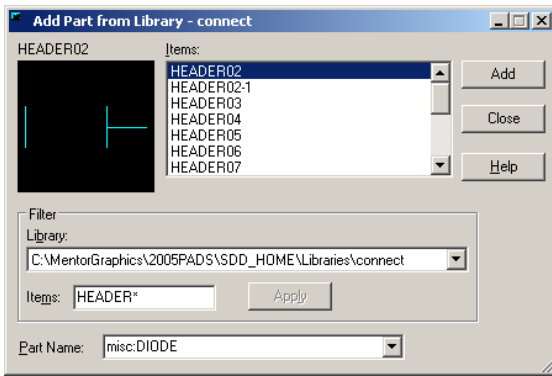
Repeat this process for the diode, selecting the DIODE2 footprint from the 'common' library.



1.9 Adding Header Pins to your design.

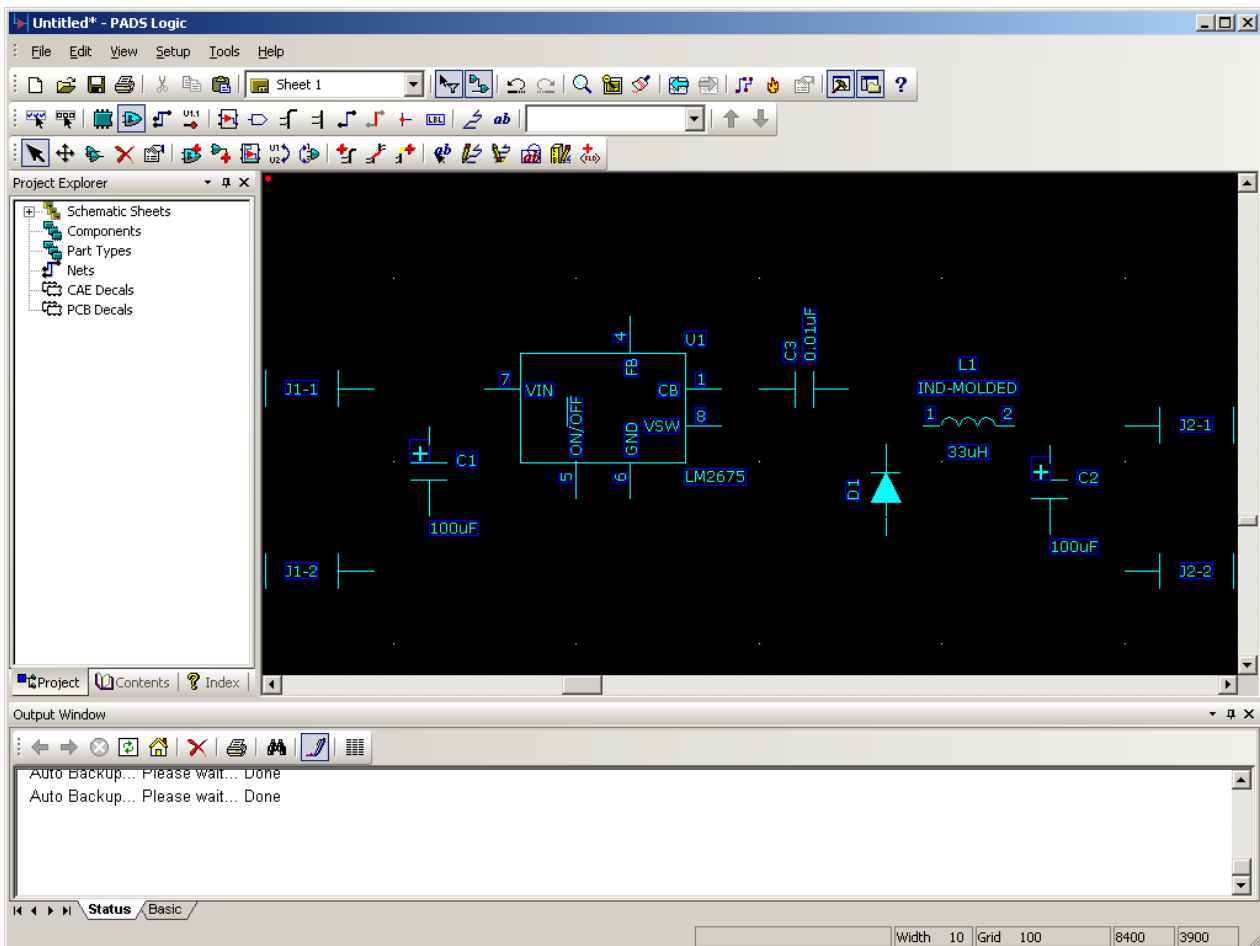
We will now add a pair of 2-pin headers to the schematic. One 2-pin header will be used for the input voltage source (12V), and the other will be used for the 3.3V output.

Open the Add Part window. Select the 'connect' library from the dropdown list and enter HEADER* in the Items filter. Press Apply and select HEADER02 from the list of parts.




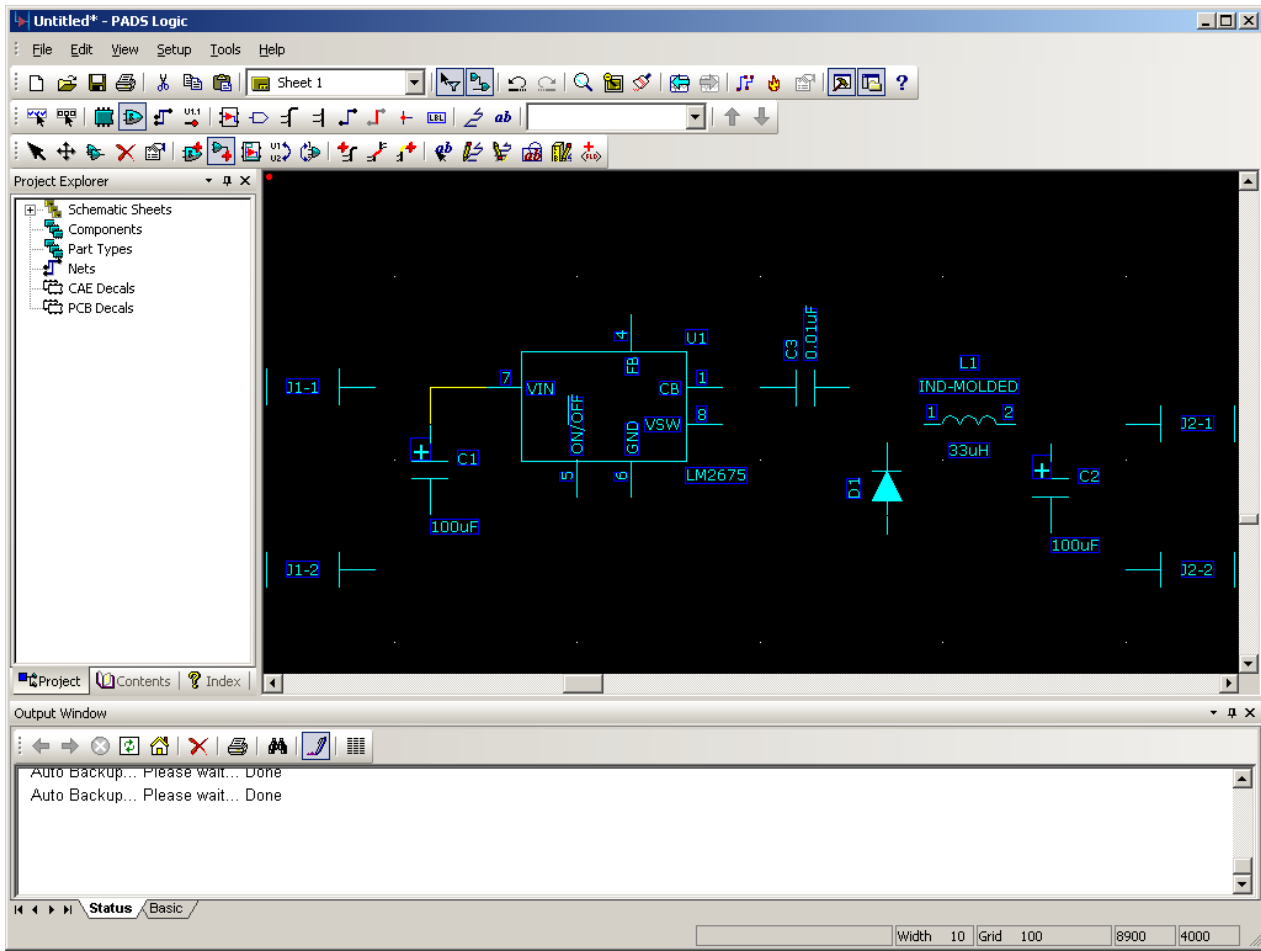
Note: PADS implements connectors as if they were multi-gate IC's. In this case, each pin is treated as a separate gate and must be added individually.

Add two header pins to the left of the schematic, and two to the right (Use **Ctrl+F** to mirror a part horizontally). Your schematic should look similar to the one below.

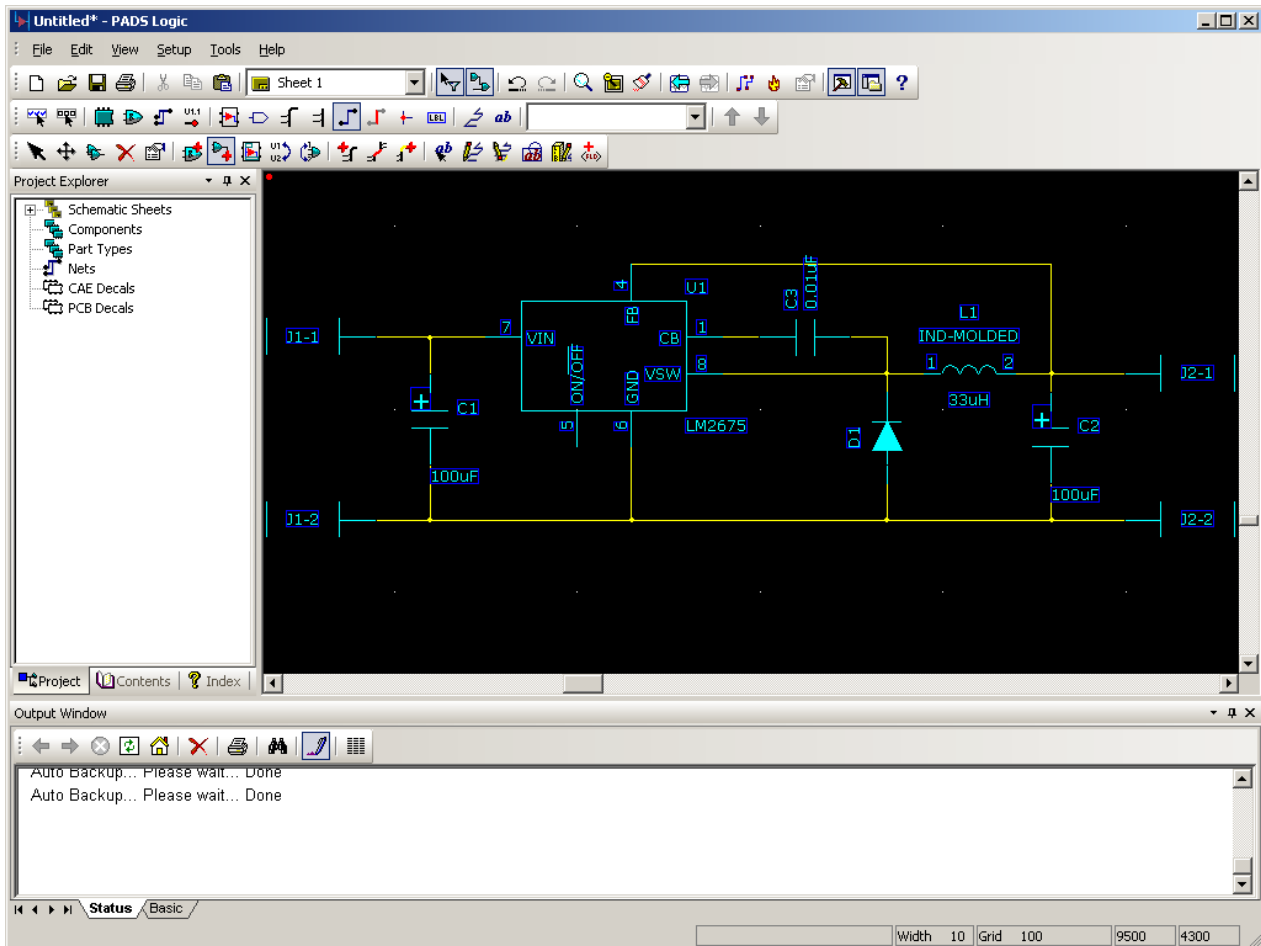



1.10 Connecting your parts.

Press the  button on the design toolbar. This will activate Add Connection Mode. Place the cursor on pin 7 of the LM2675 and click once to begin adding a connection. Move the cursor to the left until it is horizontally aligned with the left capacitor. Click once. This will add a vertex / pivot point to your connection. Move the cursor down and click on the top terminal of the left capacitor to complete the connection. Your schematic should look like the image below.



Continue adding connections until your schematic is fully connected as shown below.

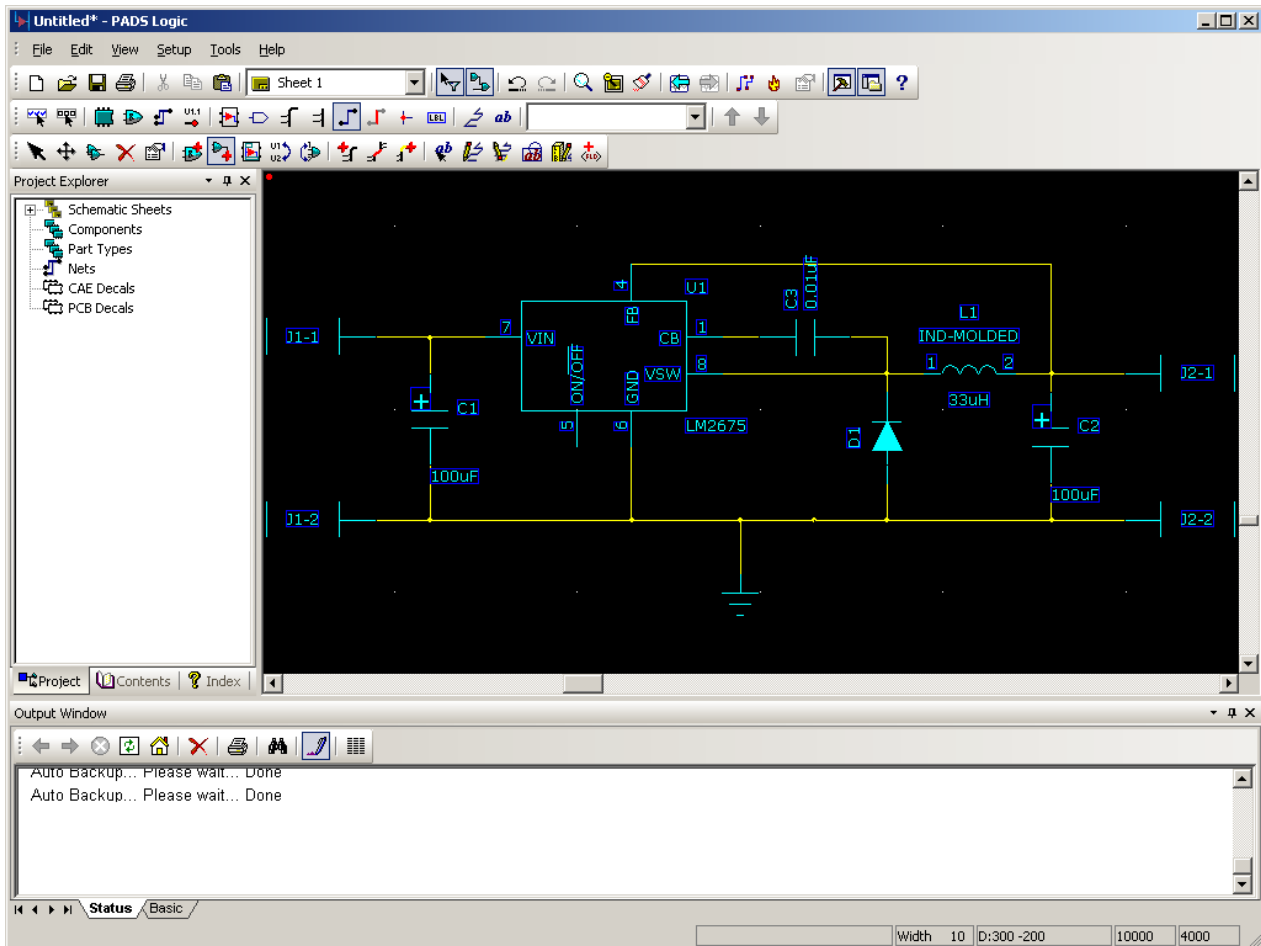


Note: To remove a misplaced connection, ensure that the selection filter allows selecting connections. Then click on the connection you wish to remove and press delete. You can also activate Delete Mode by pressing the  button on the design toolbar. This will allow you to remove objects with a single click.

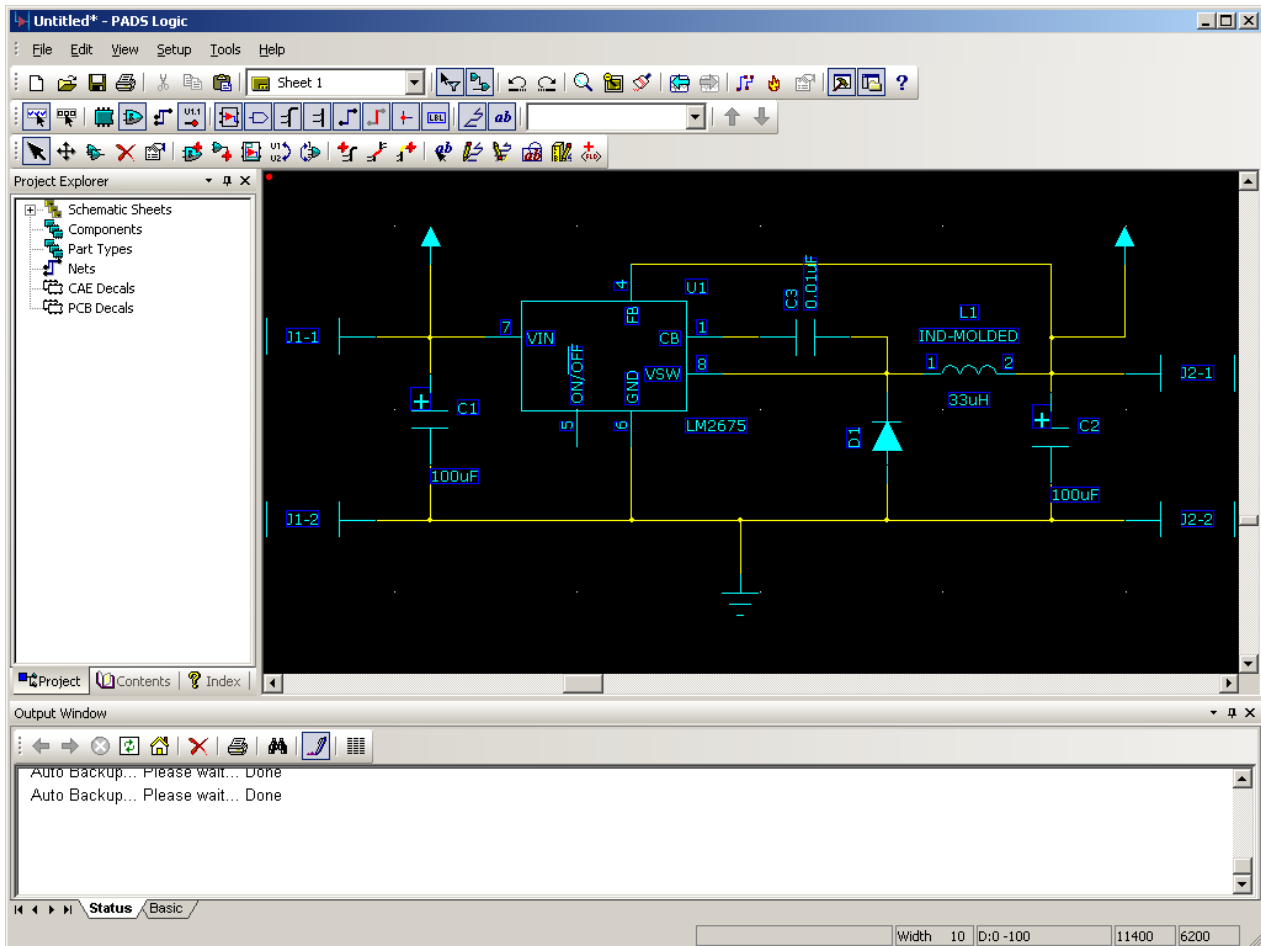
1.11 Adding Power and Ground Connections

To add a ground connection to your design, enter Connection Mode. Begin by starting a new connection from the bottom-most net and moving down 2 or 3 spaces. Then right click and select Ground from the context menu.

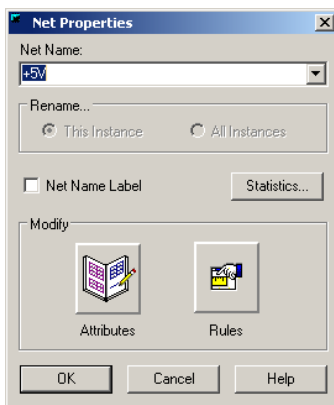
Position the ground symbol at the desired location and click once to place it.



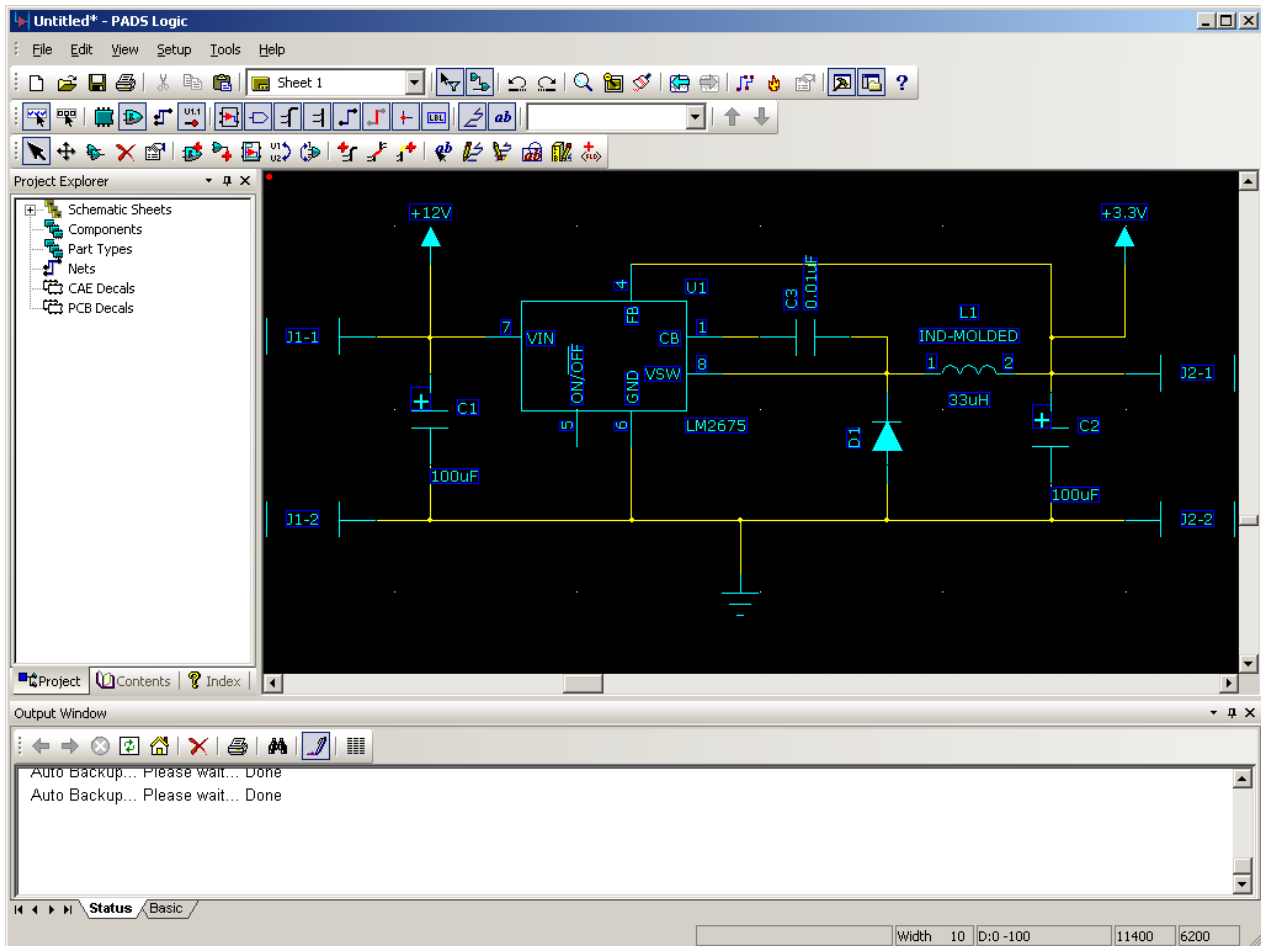
Adding a power connection is done in a similar manner. Begin by starting a new connection from the net connected to pin 7 of the LM2675 and moving up. Right click and select Power from the context menu. Position the power symbol and click once to place it. Repeat this process for the net connected to the top right header pin. Your schematic should look like the one below.



By default, all power connections are automatically connected to a net named +5V. We will need to change this for our design. Double click on the left power symbol. This will open the Net Properties window shown below.



Change the Net Name field to +12V and select the Net Name Label check box. Press OK. Repeat this process for the right power symbol, setting the Net Name to +3.3V. The completed schematic should look similar to the image shown below.



1.12 Save your schematic. Select **File > Save** and save the schematic to **H:\PADS Tutorial\Tutorial.sch**.

2. Creating a PCB Design using PADS Layout

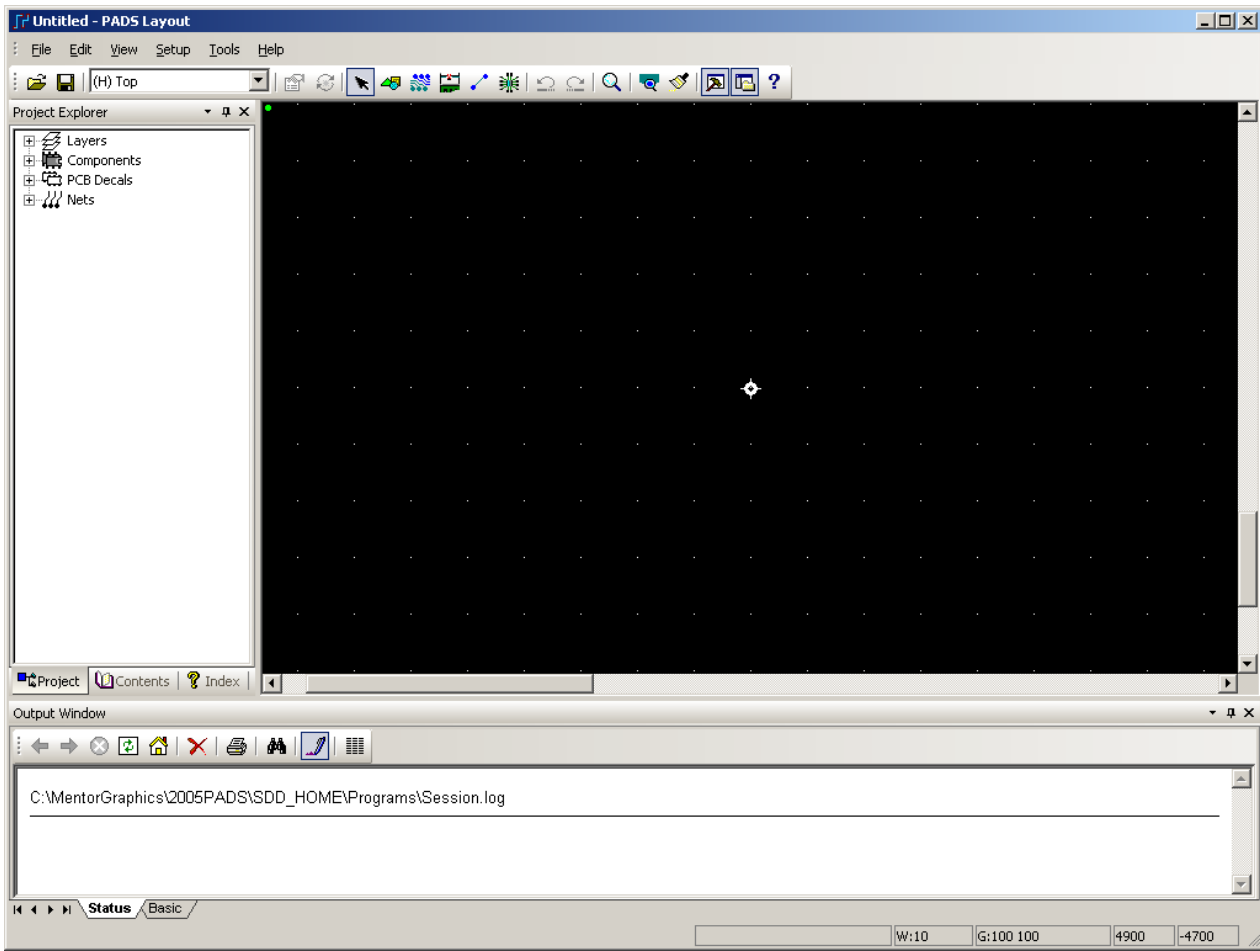
2.1 Linking PADS Logic to PADS Layout

Before you can begin laying out your PCB, you must transfer the schematic netlist to PADS Logic. The recommended way of doing this is to create an application link between PADS Logic and PADS Layout. In addition to allowing automatic synchronization of the schematic and layout information, linking also enables a feature known as cross-probing. This basically allows the selection of a part or net in PADS Logic to result in the selection of the corresponding PCB footprint or trace in PADS Layout, and vice versa.

To connect PADS Logic to PADS Layout, select **Tools > PADS Layout**. The Connect to PADS Layout dialog will appear.

Since we have not yet created a PCB Design file, click the New button. A new PADS Layout window should appear as shown below.

Note: On subsequent connections to PADS Layout or PADS Router, you will click Open and select your existing design file.



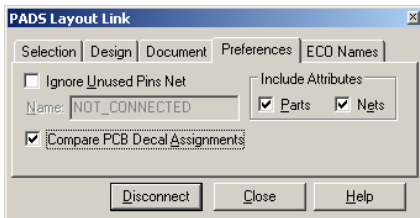
Note: If you have the available screen real estate, it is recommended that you resize the PADS Logic and PADS Layout windows so that both are visible simultaneously. A multi-monitor setup is especially useful for this.

2.2 Transferring the Netlist from PADS Logic to PADS Layout

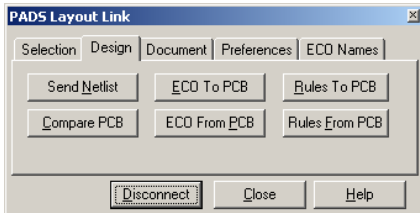
Note: Before transferring the netlist, ensure that your custom user library has been added to Layout's Library List. Refer to the New Part Tutorial if necessary.

While PADS Layout is still open, switch back to PADS Logic. The PADS Layout Link window should be visible.

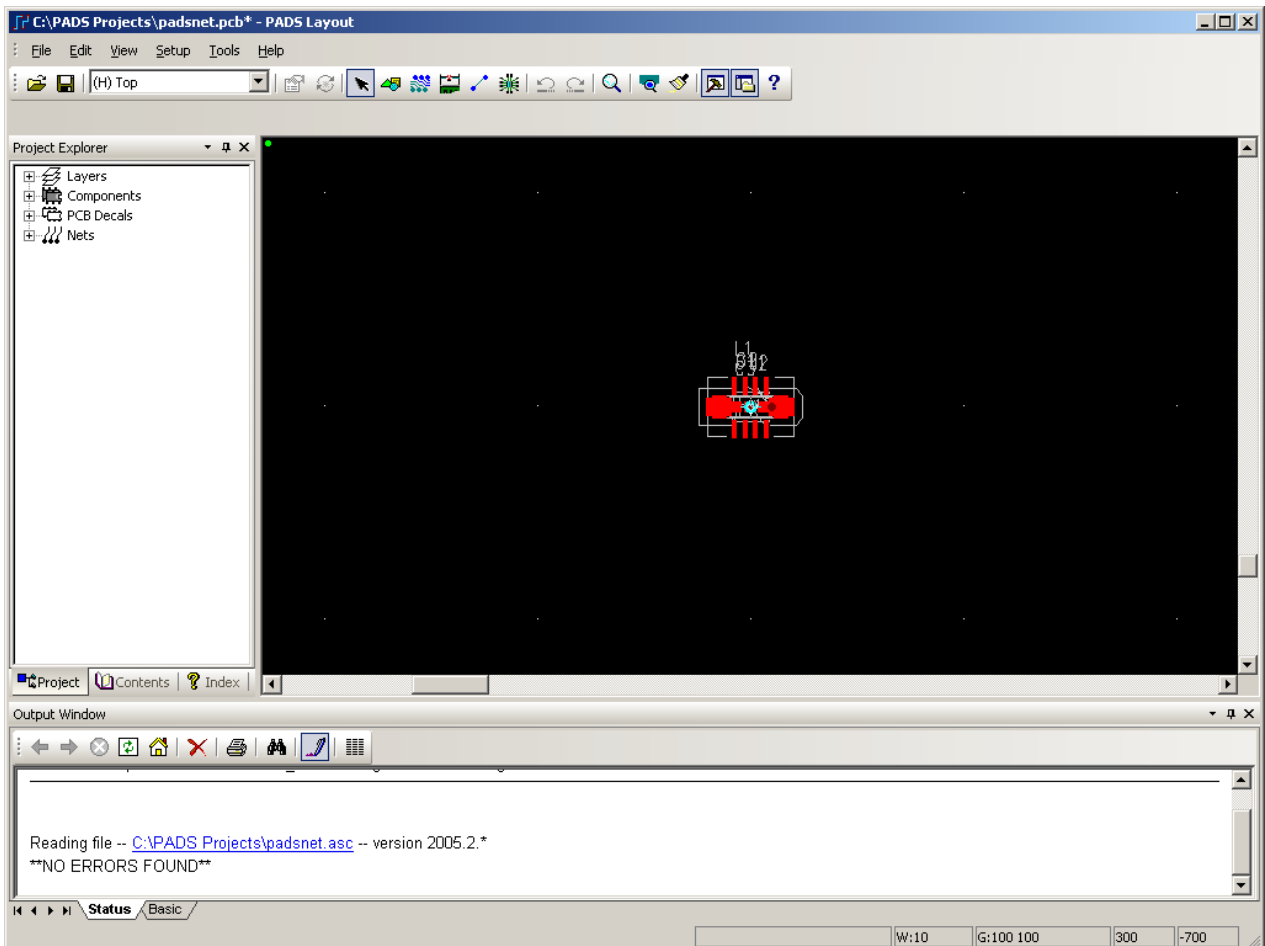
Click on the Preferences Tab and select the Compare PCB Decal Assignments checkbox. This will ensure that the modified inductor and diode footprints are properly sent to PADS Layout.




Now click on the Design Tab. Press the Send Netlist button to transfer your design to PADS Layout.



Switch back to the PADS Layout window. You should now see a cluster of parts located at the origin (Location 0,0).



2.3 Creating the board outline.

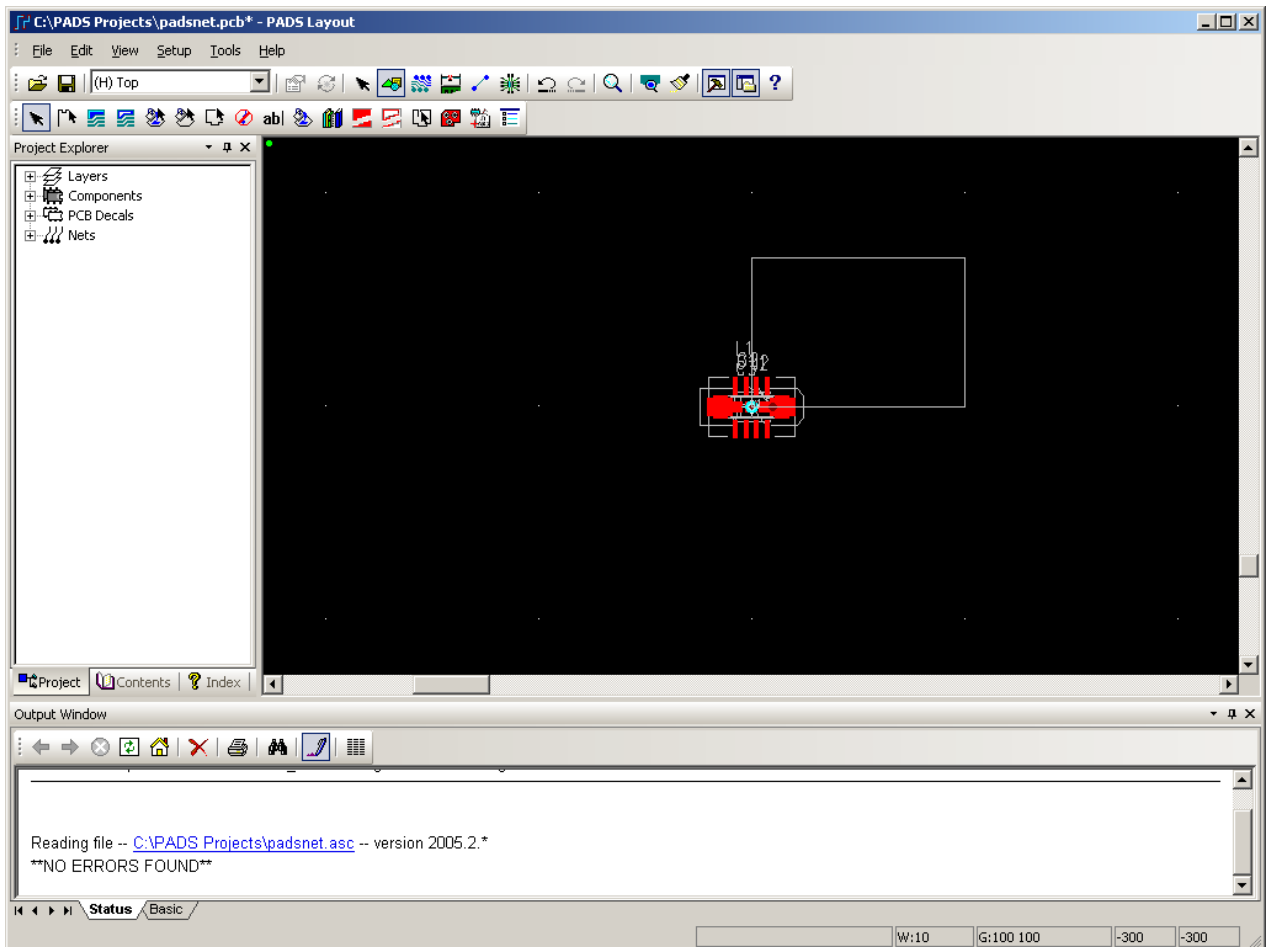
After transferring the netlist, you will need to create a board outline. Press the  button to enable the Drafting Toolbar.

On the Drafting Toolbar, click on the  button to enter board outline mode.

Since our circuit is extremely simple, a 1.0 inch by 0.7 inch PCB should be large enough. Note: For the following steps, refer to the location indicator at the bottom-right corner of the window. Locations are given assuming that design units are in mils.

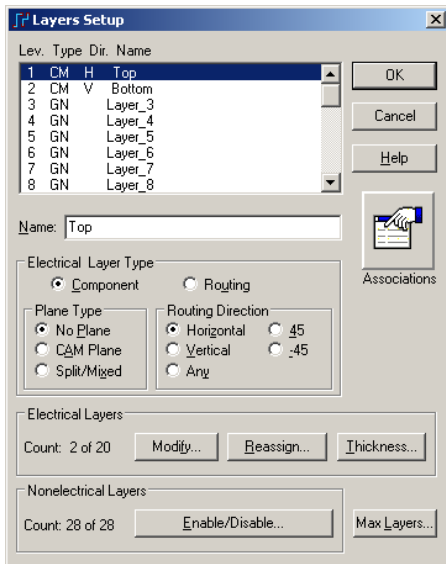
Place the cursor at the origin (location 0, 0) and click once to begin drawing the outline. Move the cursor up to location 0, 700 and click once to form the left edge of the PCB. Move the cursor right to location 1000, 700 and click once to form the top edge of the PCB. Move the cursor down to location 1000, 0 and double click to complete the PCB outline (forming the right edge and bottom edge of the PCB).

Your PADS Layout window should look similar to the one shown below.

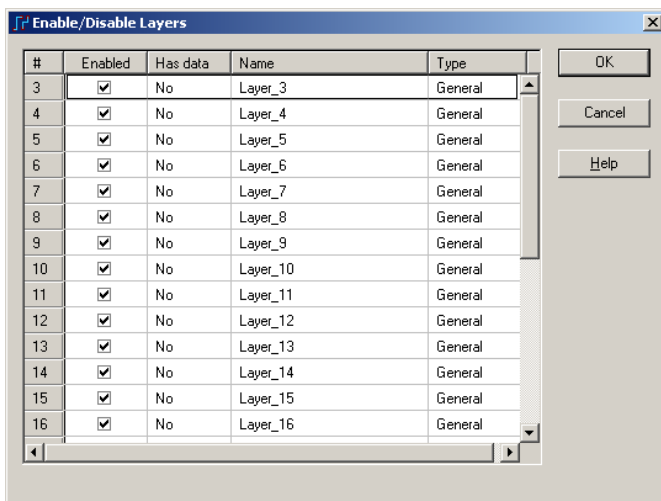


2.4 Setup the Board Layers

Before placing the components on the board, you should modify the board's Layer Definitions. Select **Setup > Layer Definition...** to open the Layers Setup dialog.



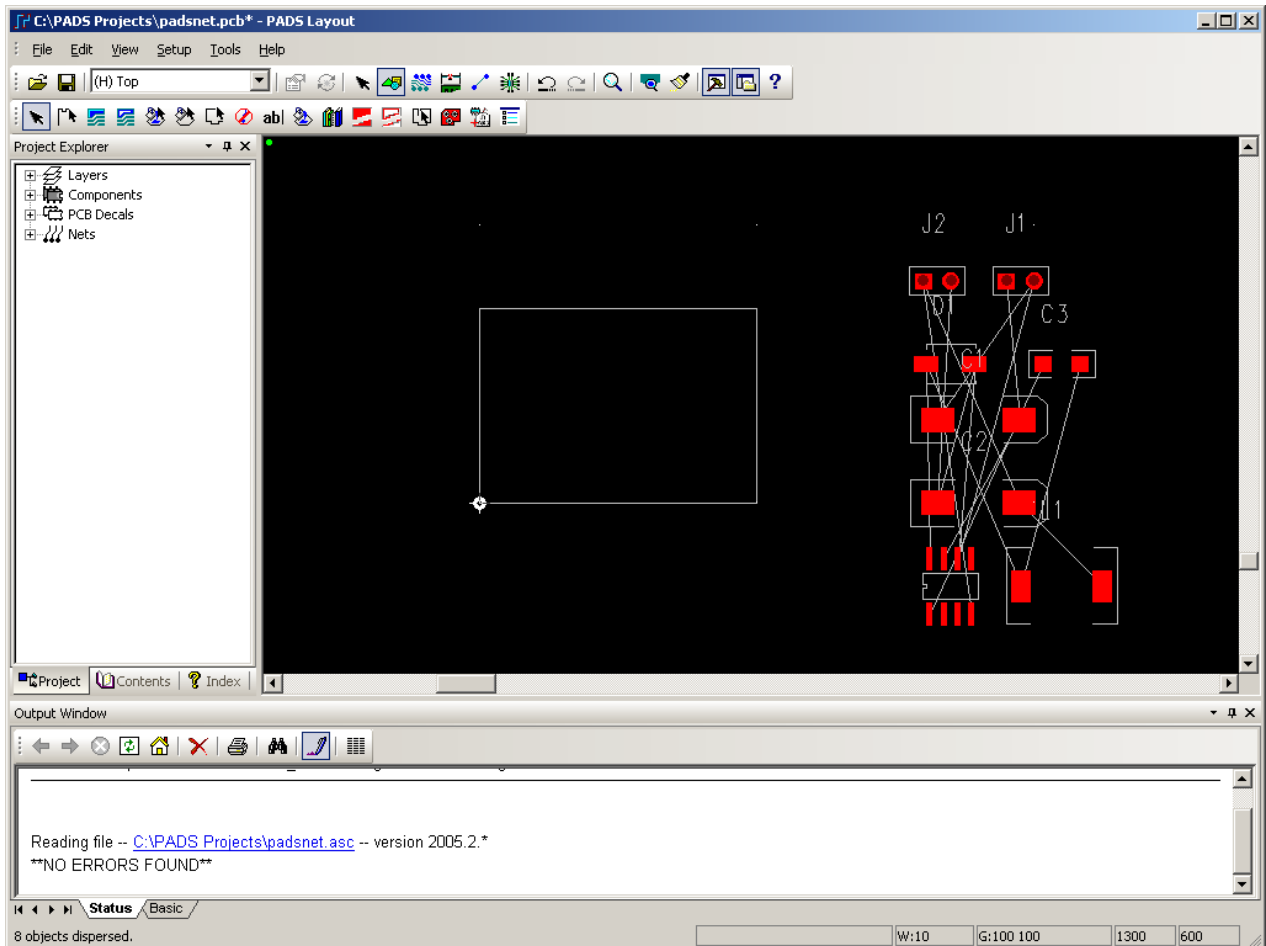
Our board will only need two electrical layers. If more than two electrical layers are assigned, press the **Modify...** button and enter **2** in the popup window to fix this. It is also recommended that you disable any unused layers. Click on the **Enable/Disable...** button to display the Enable/Disable Layers window.



Uncheck layers 3 to 20 and layer 25. Press OK to return to the Layer Setup window and then press OK again to confirm your changes.

2.5 Positioning circuit components

To make it easier to select individual parts, select **Tools > Disperse Components**. This will reposition the components around the perimeter of the board, as shown below.



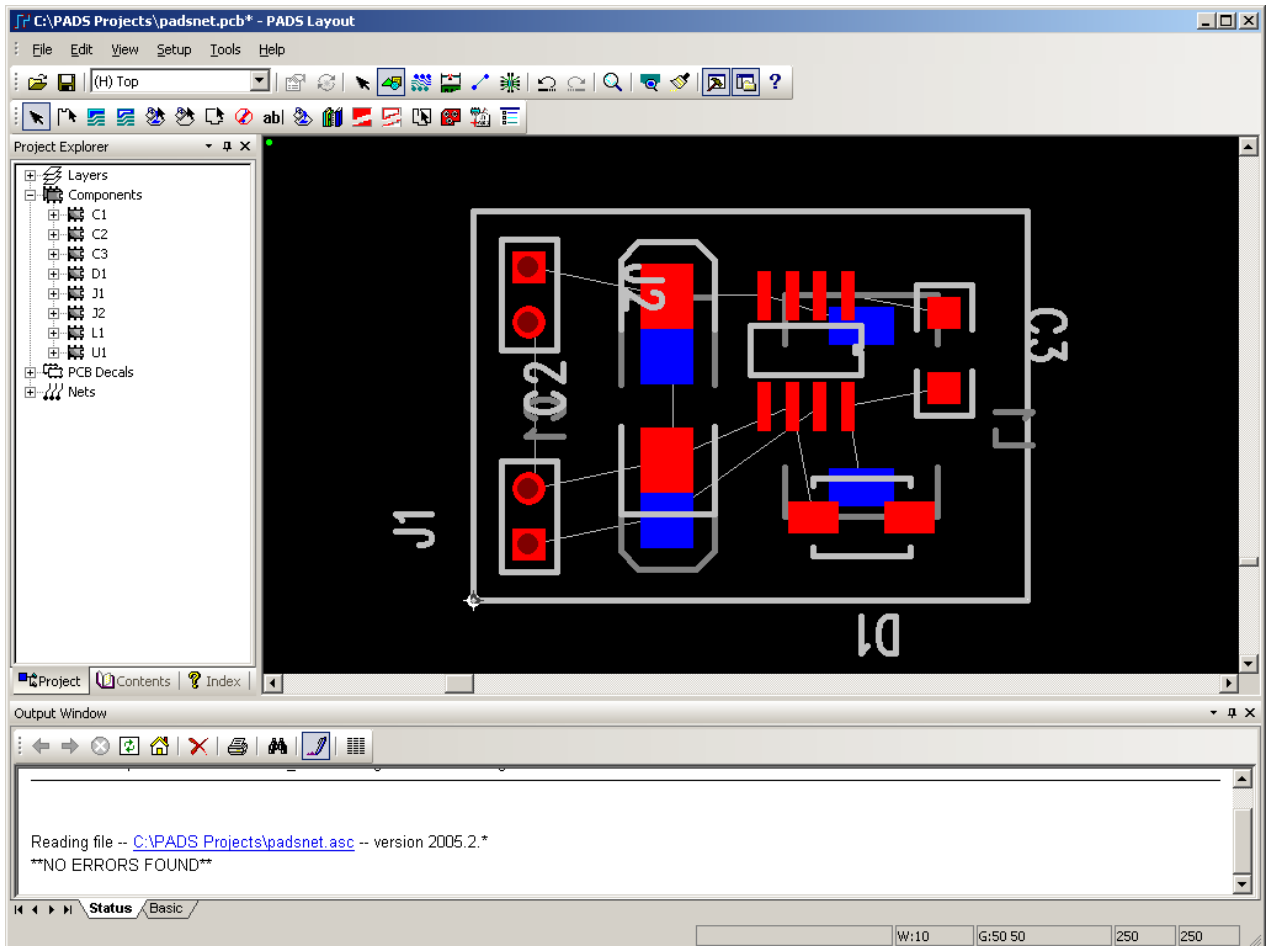
With nothing selected, right-click in an empty area and select the Select Components option from the context menu.

You can now begin placing the components within the board outline using drag and drop.

To rotate components, press Ctrl + R.

To flip a component to the opposite side of the board, press Ctrl + F.

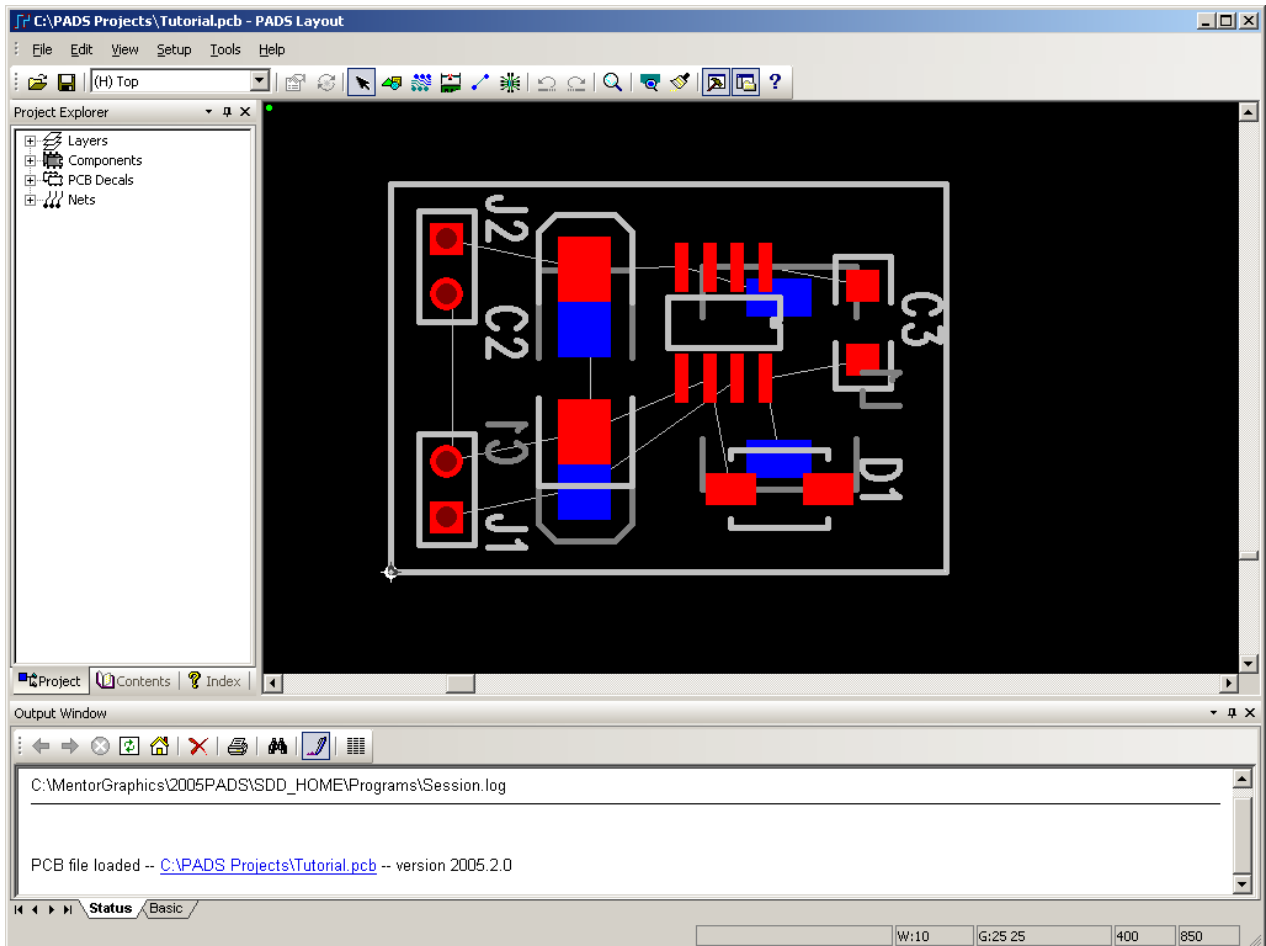
One possible layout is shown below.



Note: You may need to resize the placement grid to fit all of the parts on the board. Use the modeless grid command, G, to resize the grid. Ex: type **g 50** to set the grid interval to 50 mils.

2.6 Position the Reference Designators

You should now take the time to reposition the reference designators for your design. With nothing selected, right-click and choose **Select Documentation** from the context menu. You can now click and drag the reference designators to reposition them as shown below.



2.7 Save your PCB Layout.

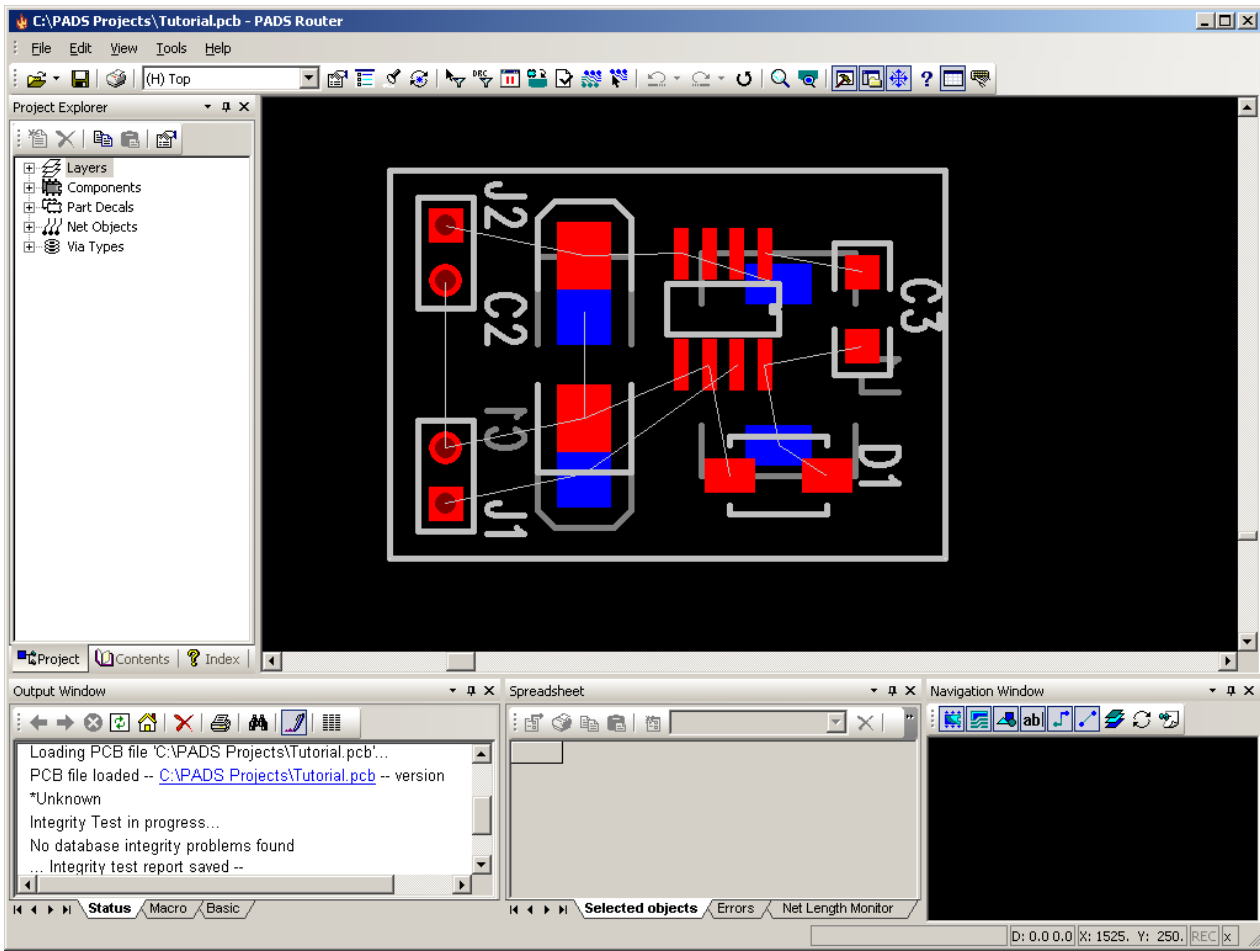
Select **File > Save** and save the file as **H:\PADS Tutorial\Tutorial.pcb**. Close PADS Layout and restore the PADS Logic window. Close the PADS Layout Link window if it is still open.

3. Routing your design using PADS Router

3.1 Linking PADS Logic to PADS Router

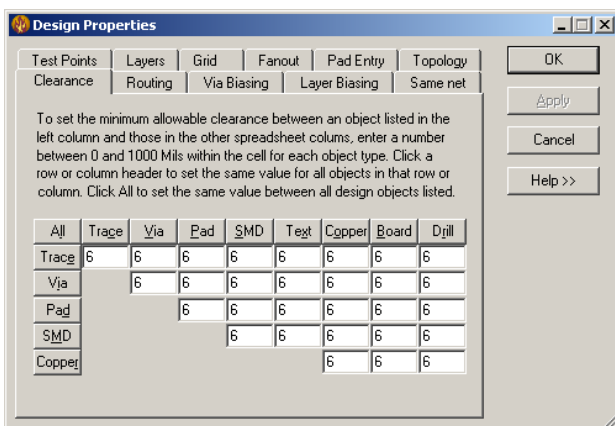
You will now connect PADS Logic to PADS Router in the same way that you connected to PADS Layout. Select **Tools > PADS Router**. The Connect to PADS Router window will appear.

Since we already have a PCB file, select **Open...** Then browse to **H:\PADS Tutorial** and open **Tutorial.pcb**. The PADS Router program should open as shown below.

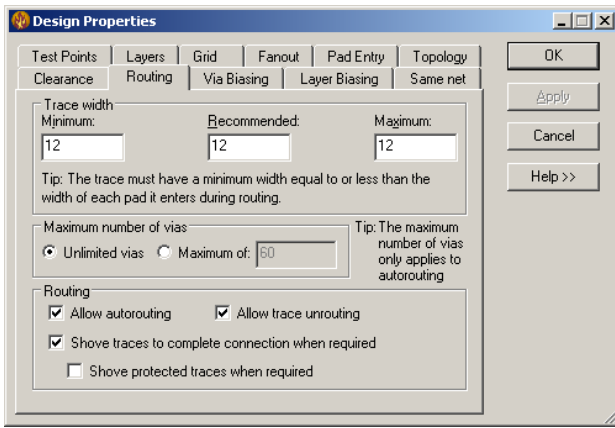


3.2 View/Modify the Design Properties

To view the current design properties, right-click and select Properties from the context menu. This will open the Design Properties window shown below.



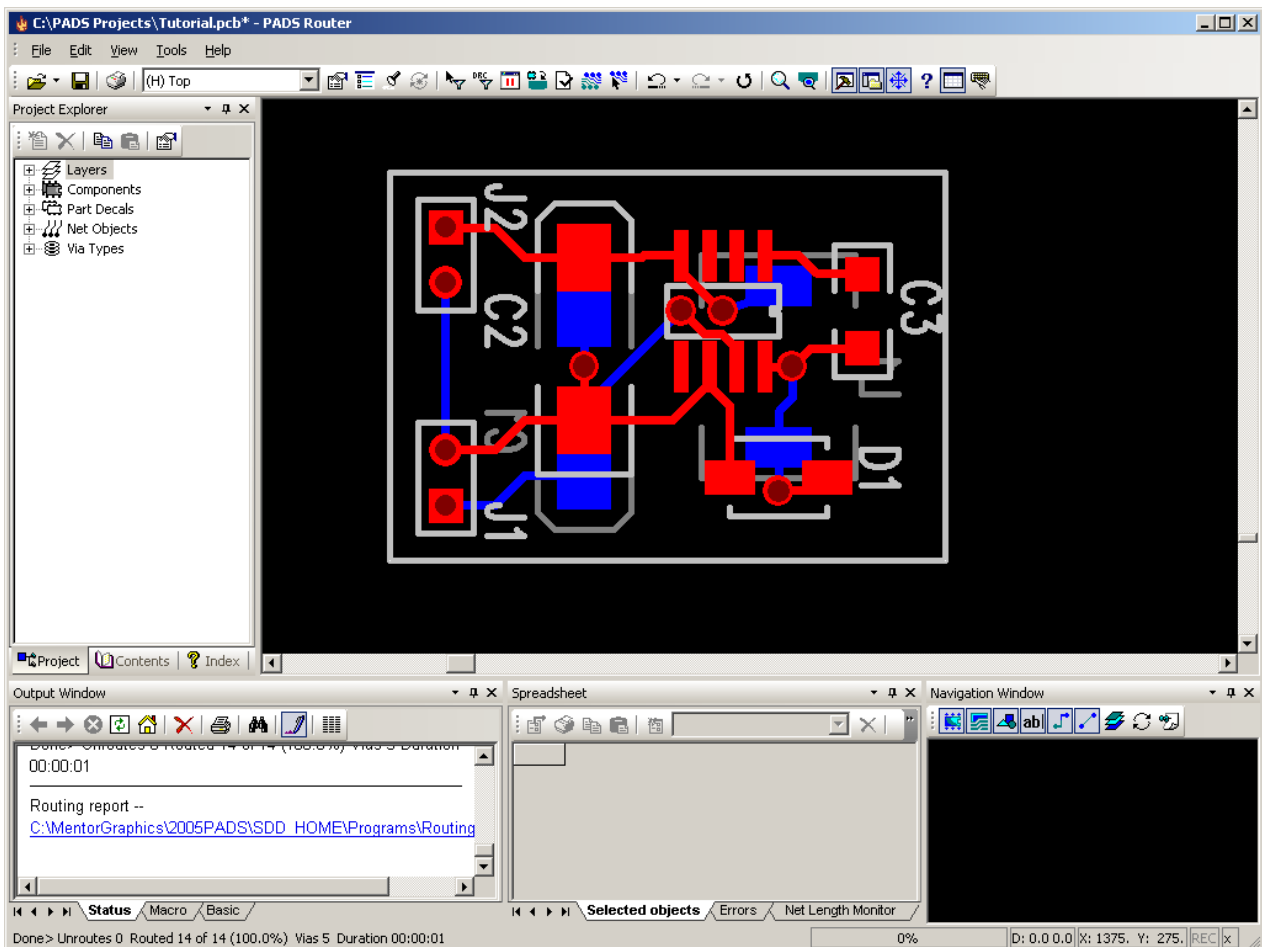
From this window you can adjust a plethora of attributes that affect your design. Select the various tabs to get an idea of all the options available. Now select the Routing tab.



From this tab you can control, among other things, the default width of your PCB's traces. Change the Recommended value to 15 and the Maximum value to 20. Then press OK to confirm your changes.

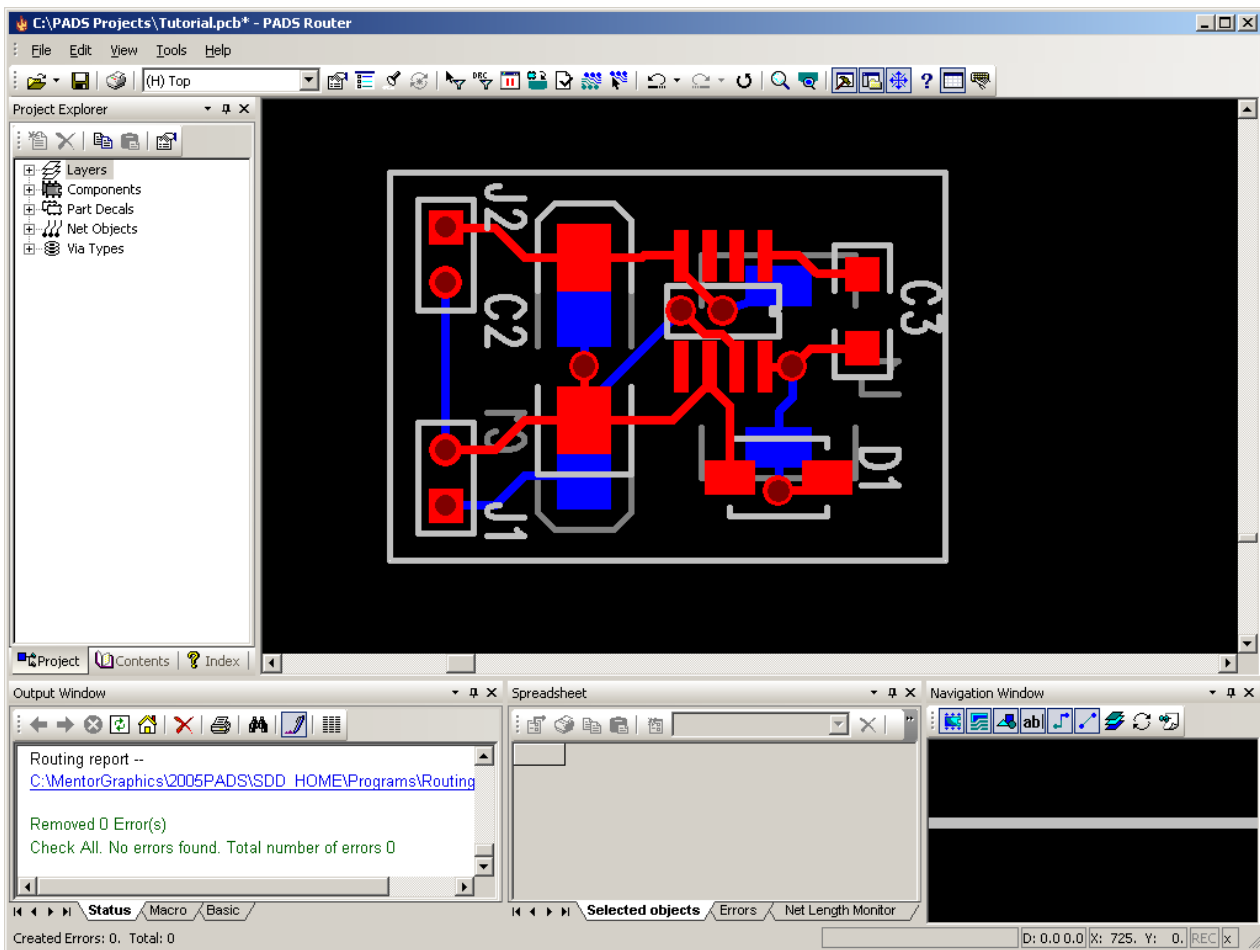
3.3 Use the PADS AutoRouter to route your design.

We will now use the built-in autorouter to route the pcb design. Select **Tools > Autoroute > Start** to begin the autoroute process. After a few seconds, the design should be fully routed as shown below.



3.4 Verify your design

You should now run a design check to ensure there are no routing errors with your design. Select **Tools > Verify Design**. The results of the test will appear in the Output Window located at the bottom right corner of the screen.

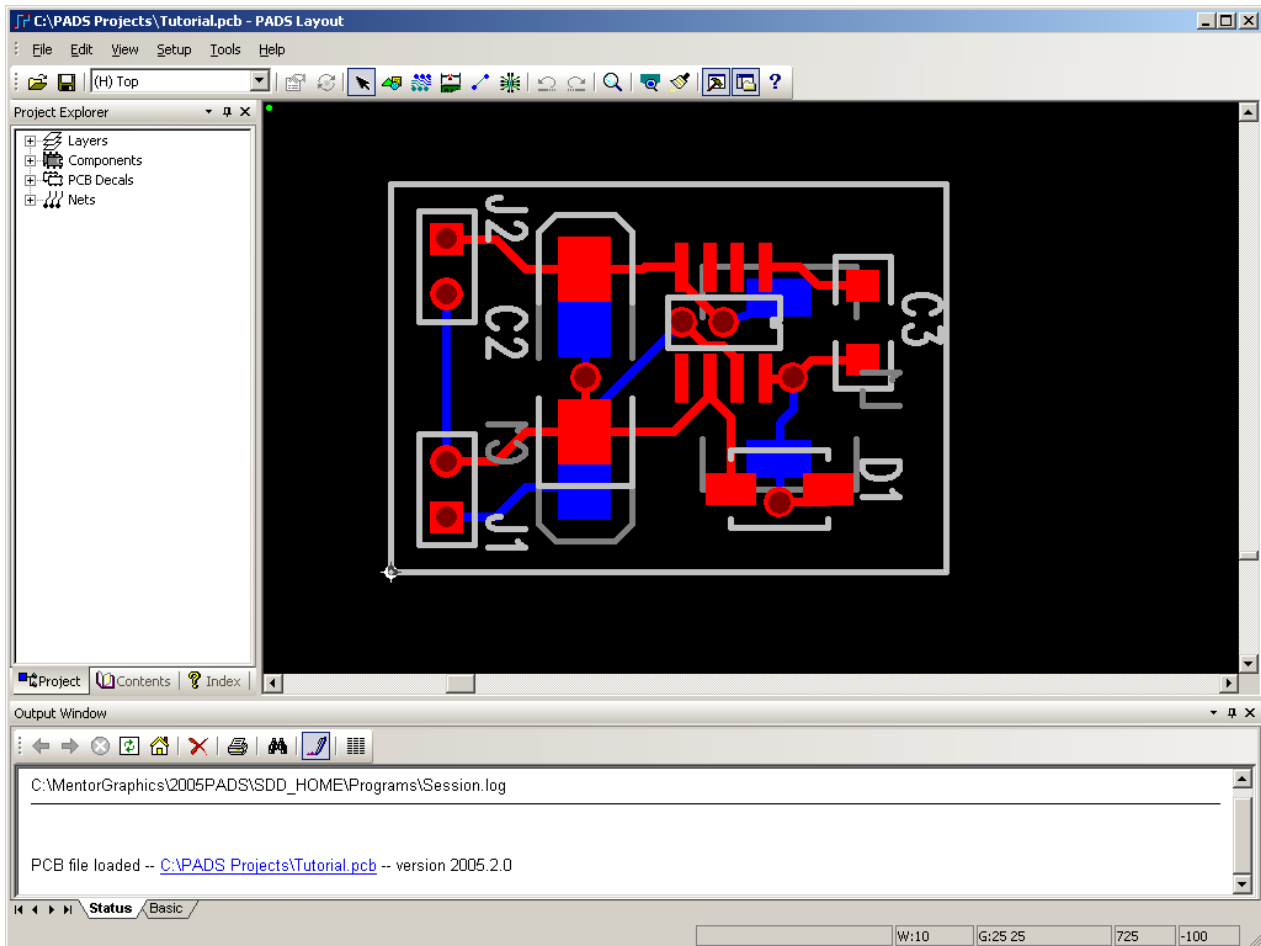


3.5 Save your design. Select **File > Save** and then close PADS Router, returning to PADS Logic. Close the PADS Router Link dialog if it is still open.



4. Finishing your design in PADS Layout

4.1 Linking PADS Logic to PADS Layout.

You will now connect to PADS Layout again and complete the final steps in your design. Select **Tools > PADS Layout** to open the Connect to PADS Layout window. Select **Open...**, browse to **H:\PADS Tutorial** and open **Tutorial.pcb**. PADS Layout should open, displaying your fully routed design.

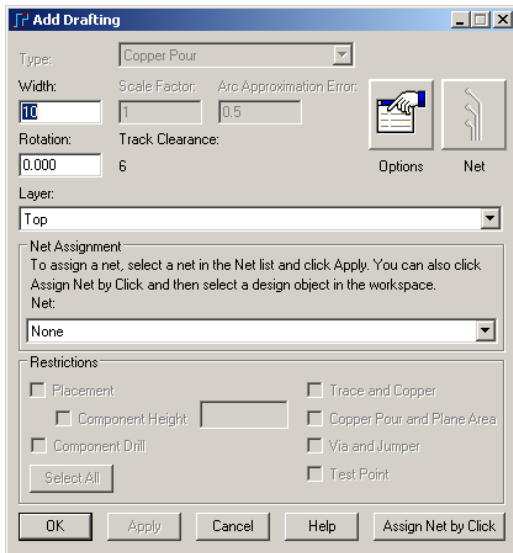


4.2 Adding Copper Pours to your PCB

You will now add a copper pour to each side of your PCB. Enable the drafting toolbar by pressing the  button. Now press the  button to enter Copper Pour Mode.

Move the cursor over the origin and click once to start drawing the outline of the copper pour.
 Move the cursor to location 0,700 and click to draw the left edge.
 Move the cursor to location 1000,700 and click to draw the top edge.
 Move the cursor to location 1000, 0 and double click to complete the copper pour outline.

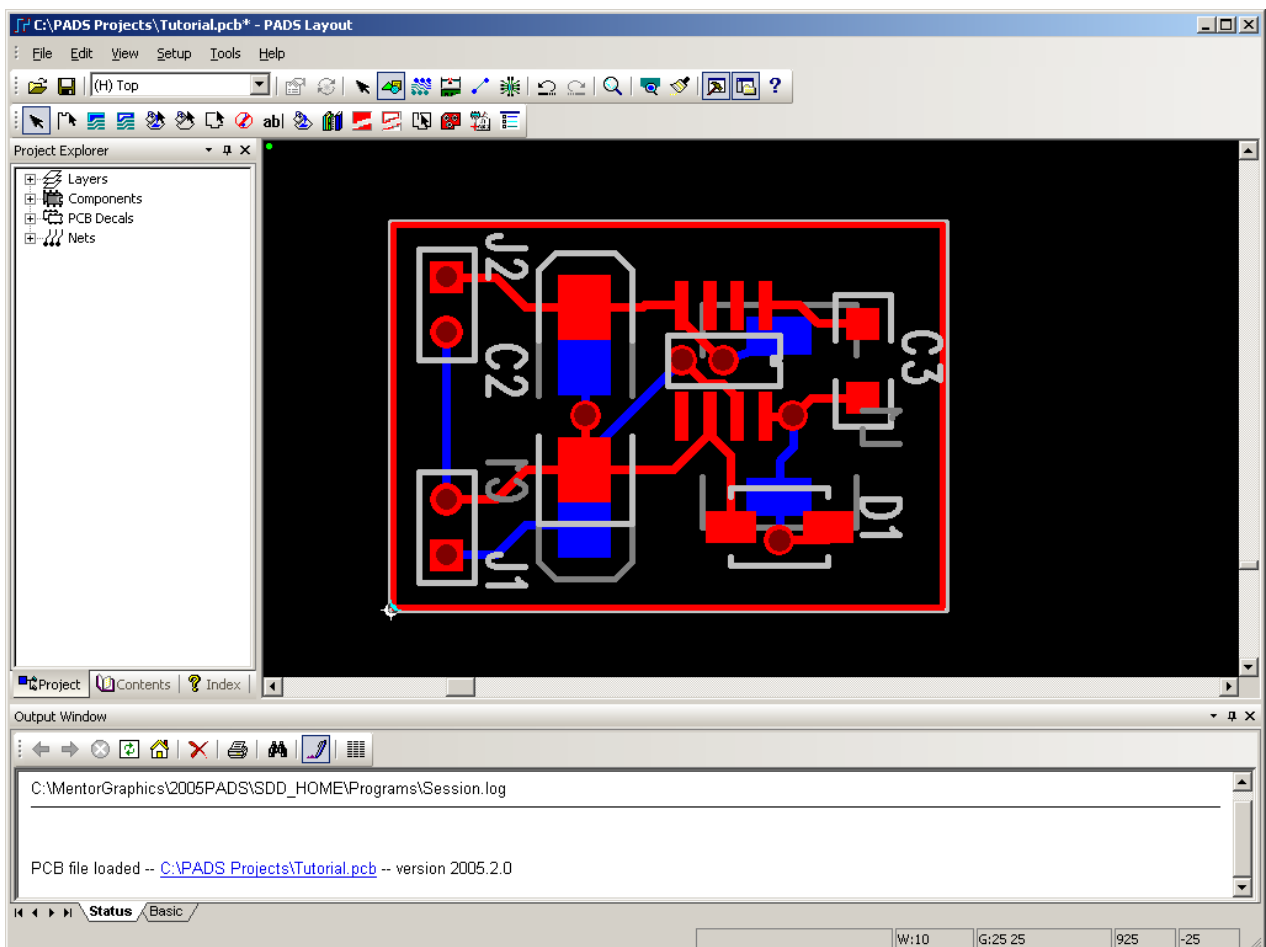
The Add Drafting should appear as shown below.



Select Top from the **Layer** dropdown list and select GND from the **Net** dropdown list. Press OK.

Repeat this process using the same coordinates and choosing Bottom for the Layer. Select GND for the Net.

After the copper pours have been added, the design window should appear as shown below.

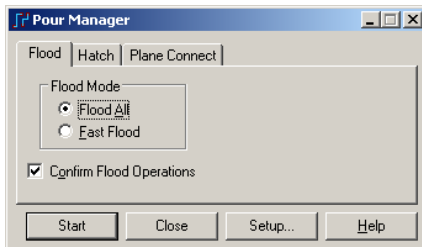


Note: If you need to re-route your design, you will need to remove the copper pours and

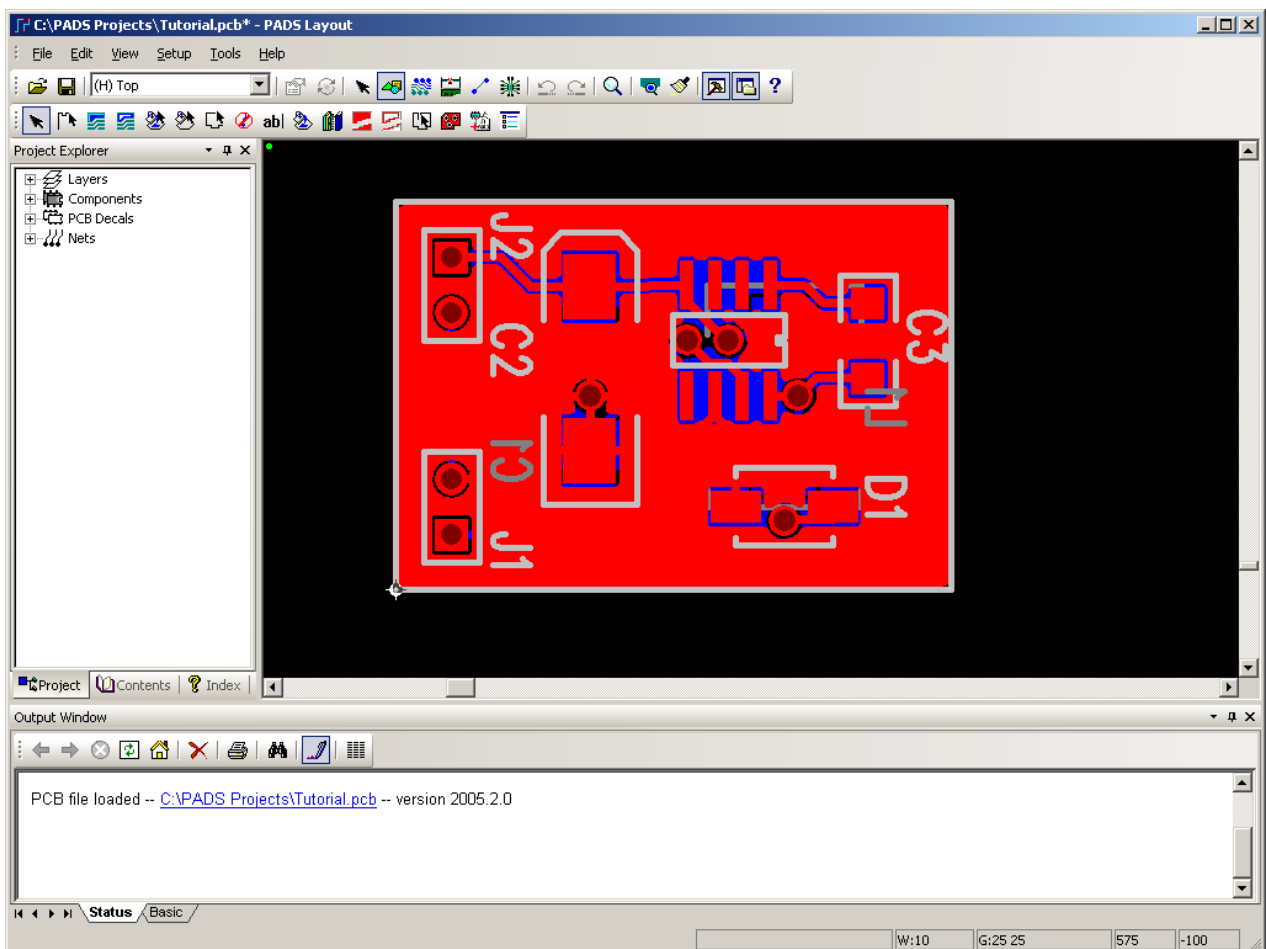
save your file before opening it in PADS Router. If you fail to do this, PADS Router may not correctly route your PCB.

4.3 Flooding your design.

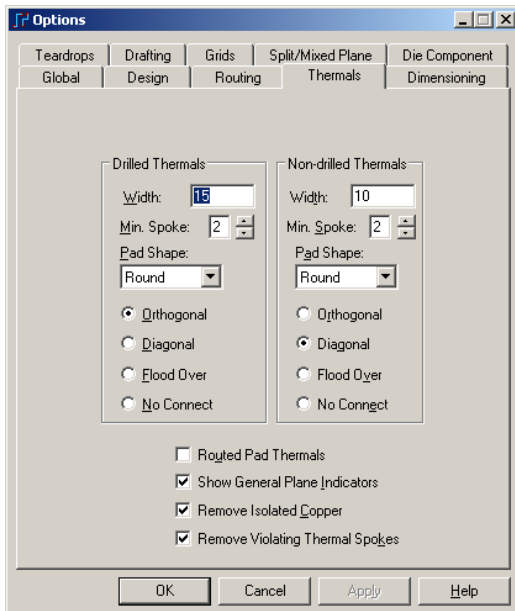
You will now flood your design, filling in the copper pours with copper. Select **Tools > Pour Manager** to open the Pour Manager window.



Press **Start** to begin the flood process. If there were no errors, close the Pour Manager. Your design window should now look similar to the image below.



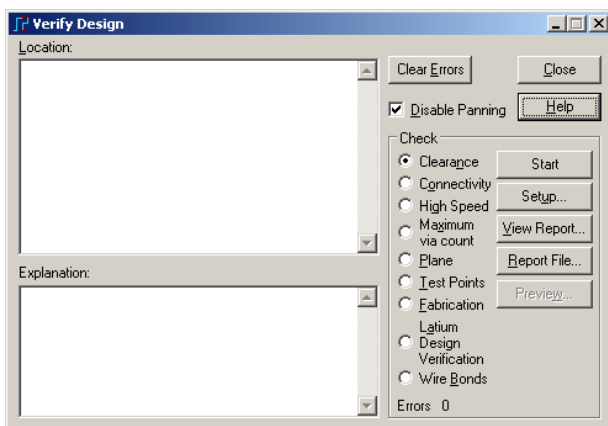
If there were any errors, press the **Setup** button to open the Options > Thermals window.



Any copper pour errors can often be corrected by changing the type or width of the thermal spokes. Changing the type to 'Flood Over' should always prevent the 'too few spokes generated' error. This, of course, changes the method used to connect the copper pour to vias and pads of the same net. Note: Not all copper pour errors are critical, it may not be deemed necessary to modify the Thermals parameters.

4.4 Verify Design

You should now check your completed design for errors. Select **Tools > Verify Design** to open the Verify Design window.



Select Clearance from the Check list and press start. If there are no errors, the following dialog will be displayed.

Press OK and select Connectivity from the Check list. Press Start to run the check. If there were no errors, close the Verify Design window. If any errors were detected, you should

locate the error image on your design and correct the problem.

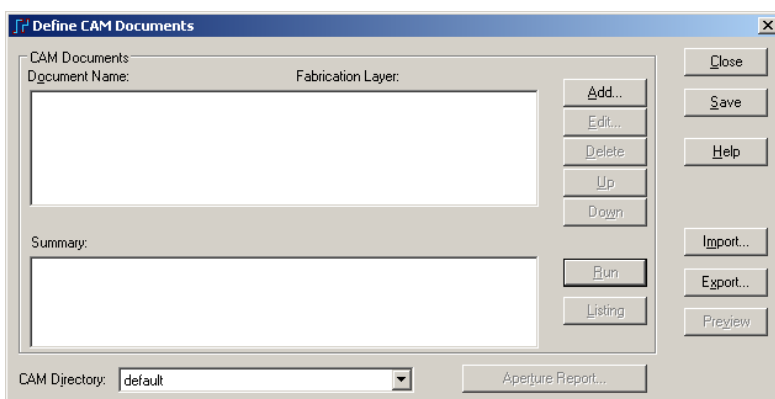
Note: If you need to re-route your design, you will need to remove the copper pours and save your file before opening it in PADS Router. If you fail to do this, PADS Router may not correctly route your PCB.

4.5 Save your design. Select **File > Save** to save your completed PCB design.

5. Generating Gerber Files

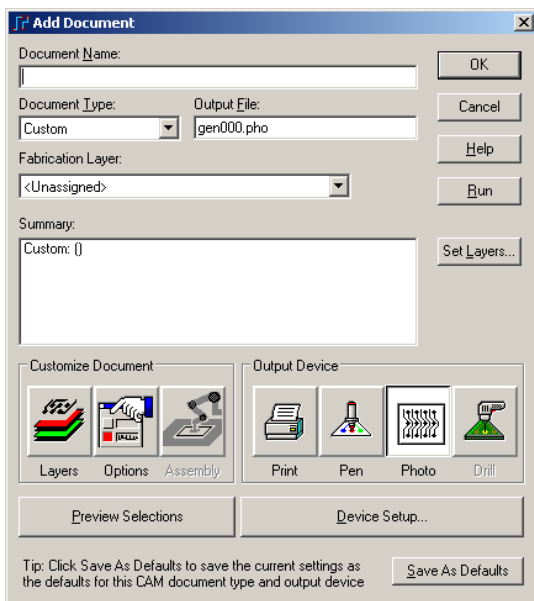
The steps needed to generate the appropriate gerber files will vary depending on the number of layers used and the PCB service that will be creating the board. This section will show the steps needed to create gerber files for a PCBExpress E2 2-Layer Board.

5.1 Select **File > CAM** to open the Define CAM Documents window.



5.2 Top Layer

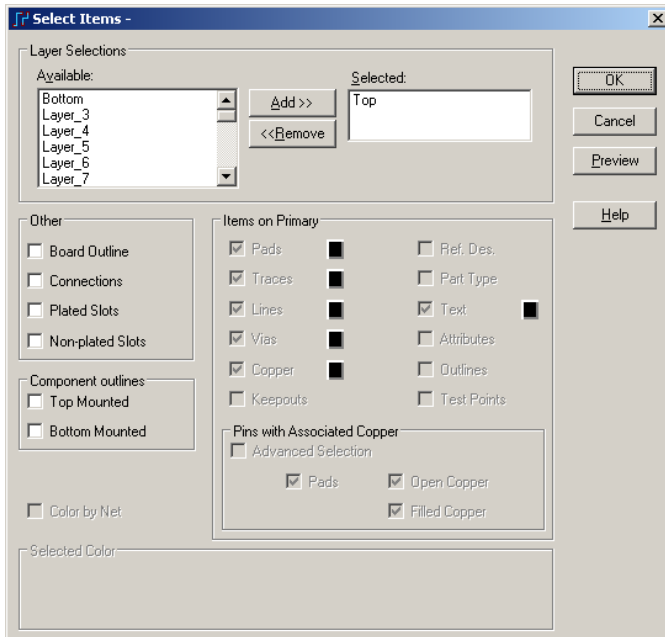
Press the **Add...** button to display the Add Document window.



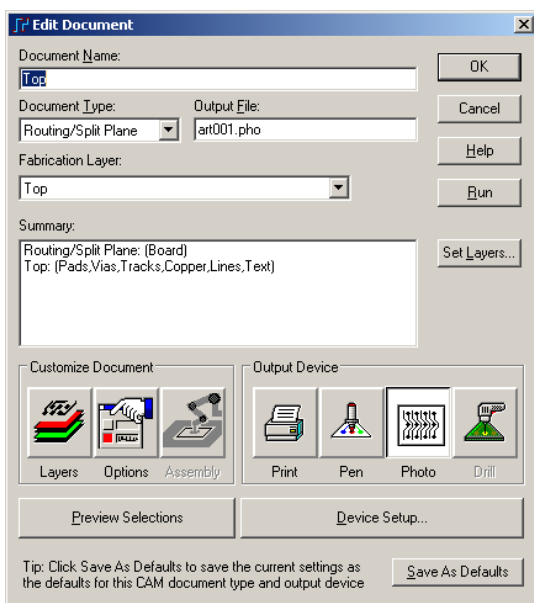
Type **Top** in the Document Name field.

Select **Routing/Split Plane** from the Document Type dropdown list and select **Top** from the popup window.

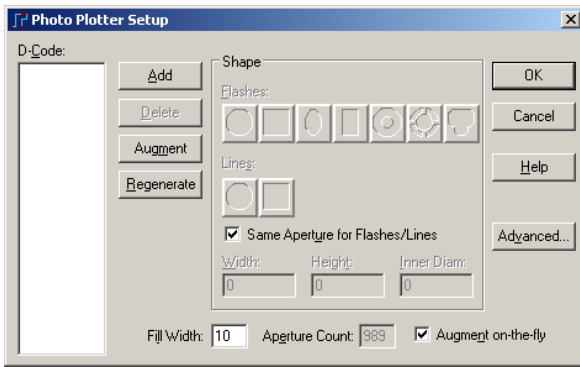
Press the **Layers** button to open the Select Items window.



Check the **Board Outline** checkbox and press OK. The Add Document window should look like the one below.

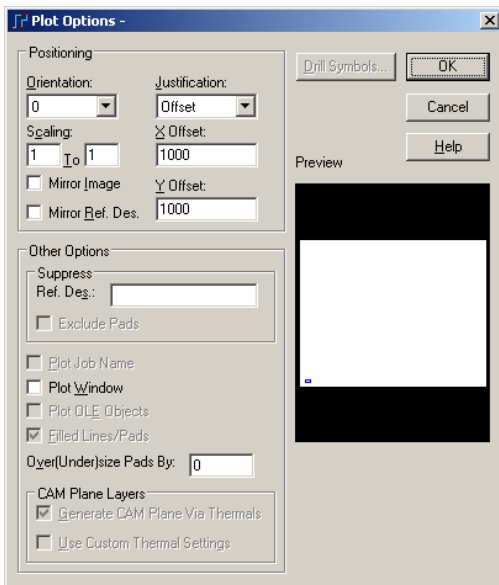


Press the **Device Setup** button to display the Photo Plotter Setup window.



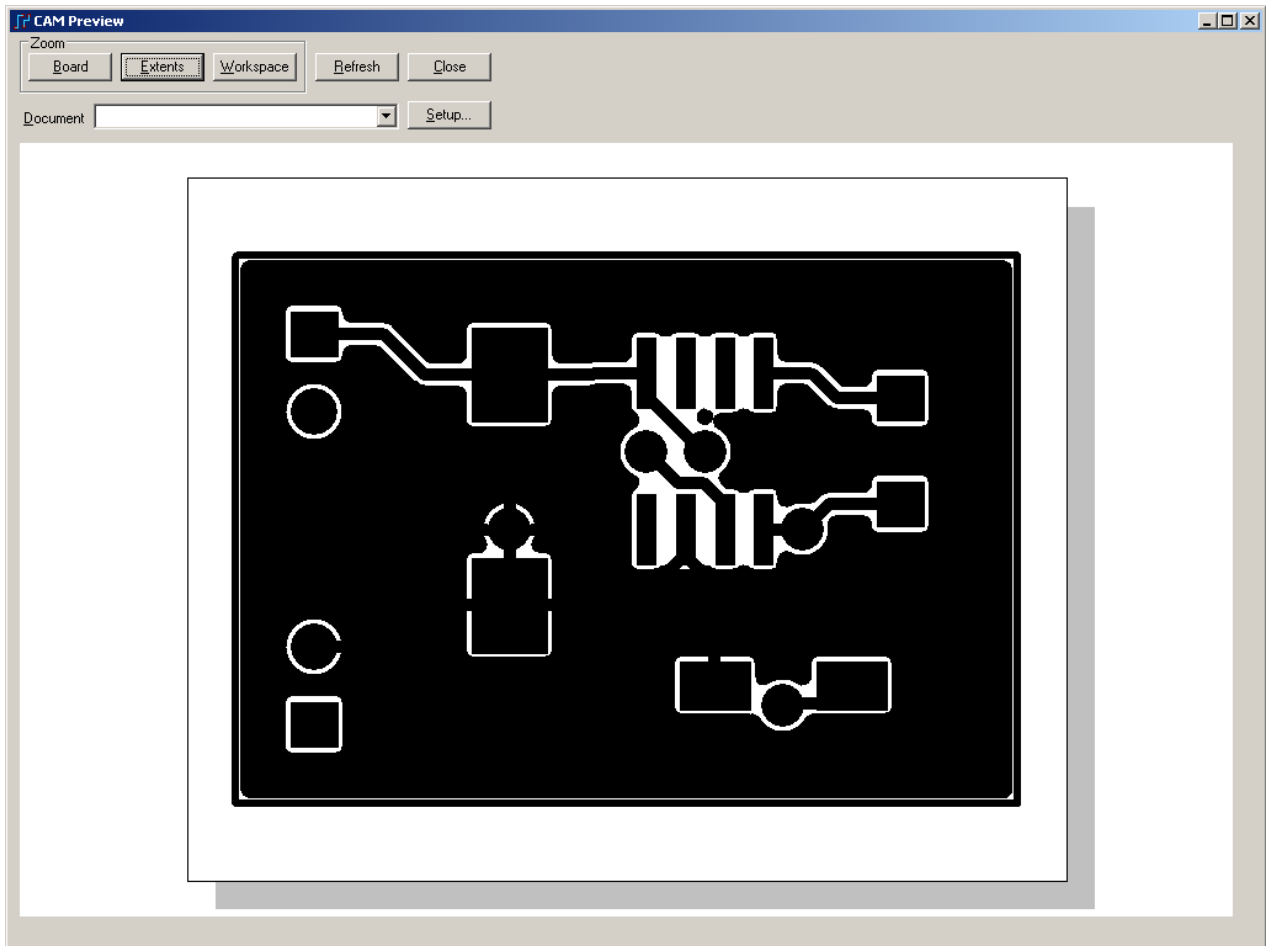
Press the **Regenerate** button and click yes. Press OK.

Press the **Options** button to display the Plot Options window.

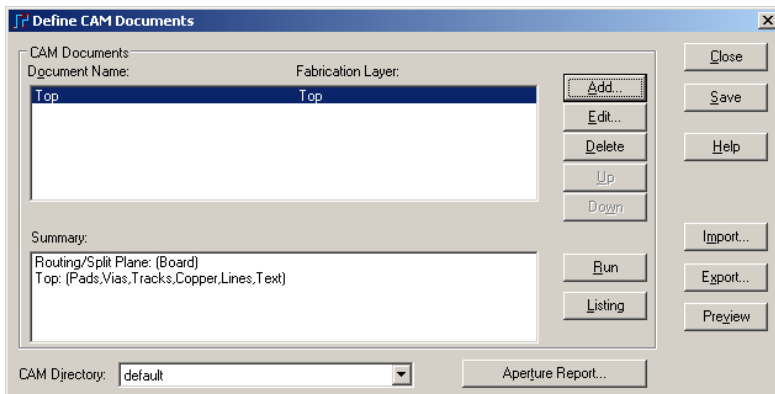


Change the Justification setting to **Scale to Fit** and press OK.

Press the **Preview Selections** button on the Add Document window to verify your settings.

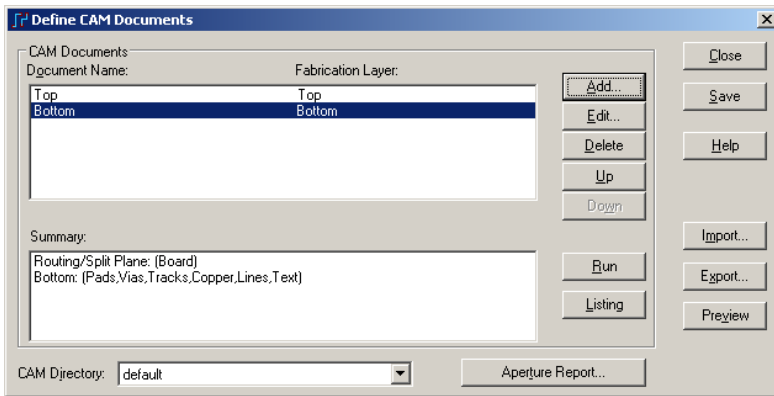


If everything looks correct, close the preview window and then press OK to add the Top document.



5.3 Bottom Layer

Repeat part 5.2, substituting the **Bottom** Layer for the Top Layer in each step.



5.4 Silkscreen Top

Press the **Add...** button.

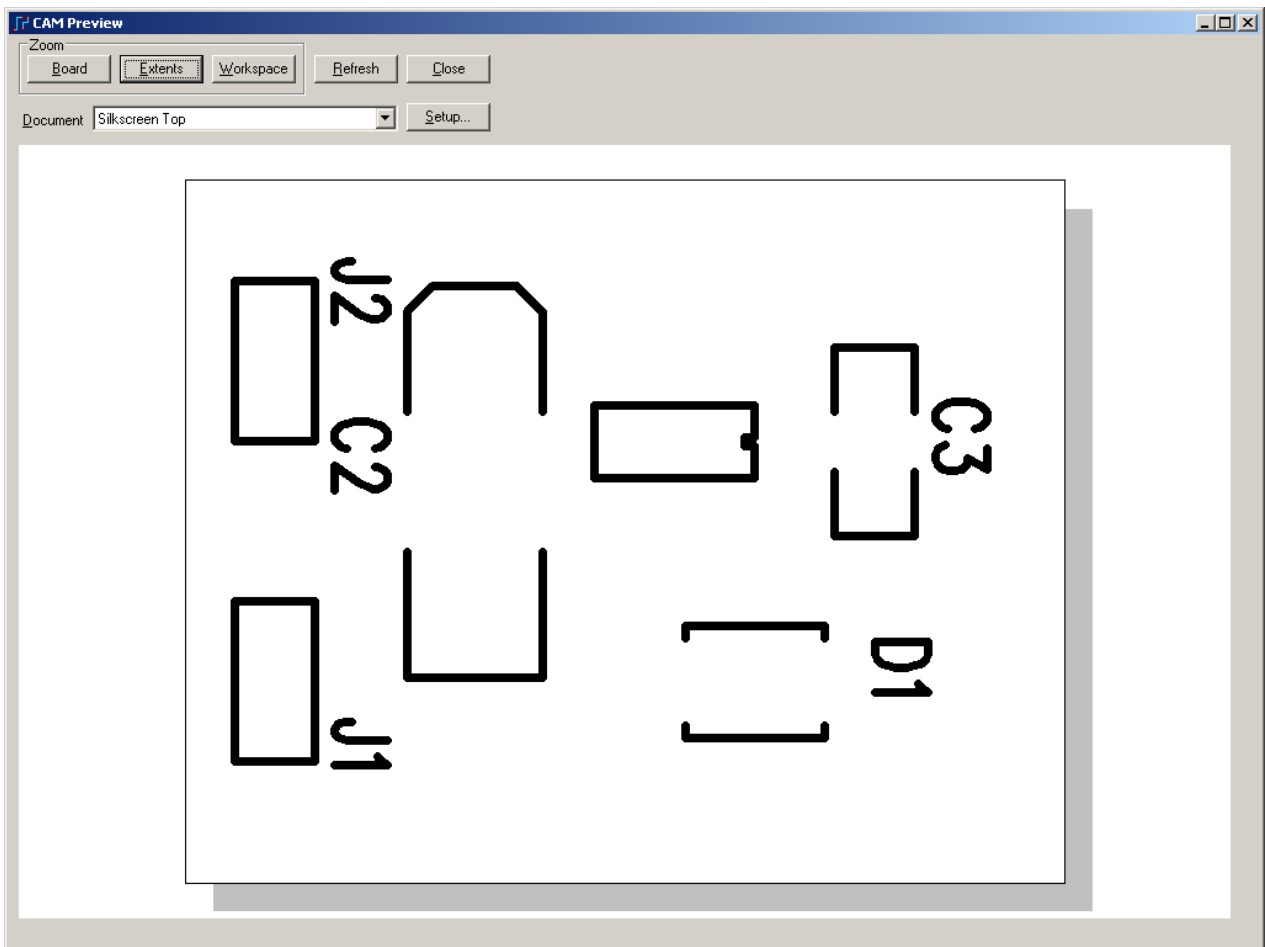
Type **Silkscreen Top** for the Document Name.

Select **Silkscreen** from the Document Type list and then select **Top** from the popup window.

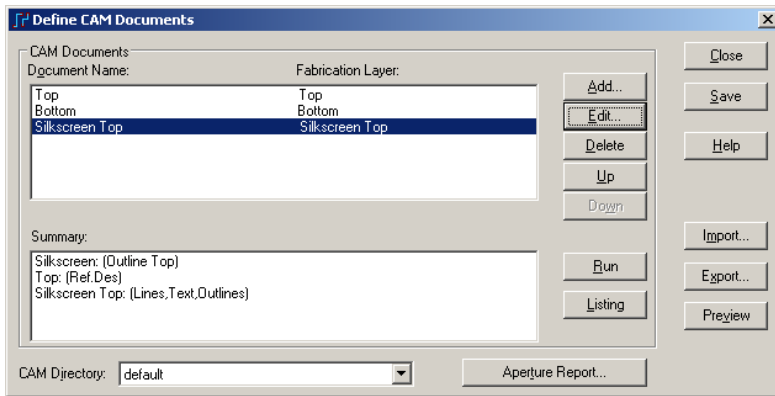
Press the **Layers** button to open the Select Items window. Select **Top** from the Selected List and uncheck **Part Type**. Press OK.

Press the **Options** button and set the Justification to **Scale to Fit**. Press OK.

Press the **Preview Selections** button to verify your design.

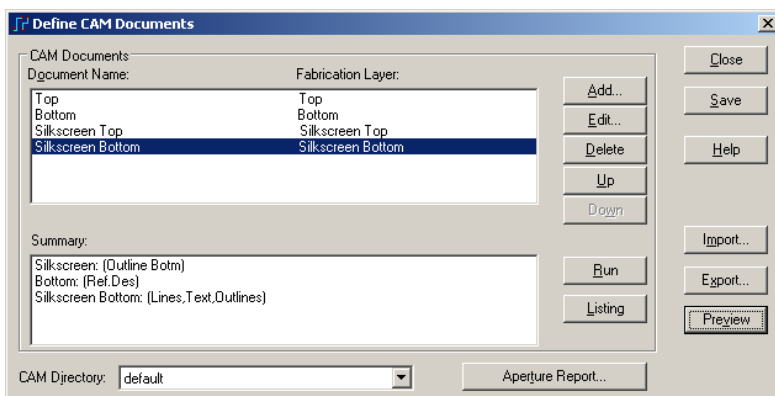


Close the preview and press OK to add the Silkscreen Top document.



5.5 Silkscreen Bottom

Repeat Section 5.4, substituting Silkscreen Bottom for Silkscreen Top.



5.6 Solder Mask Top

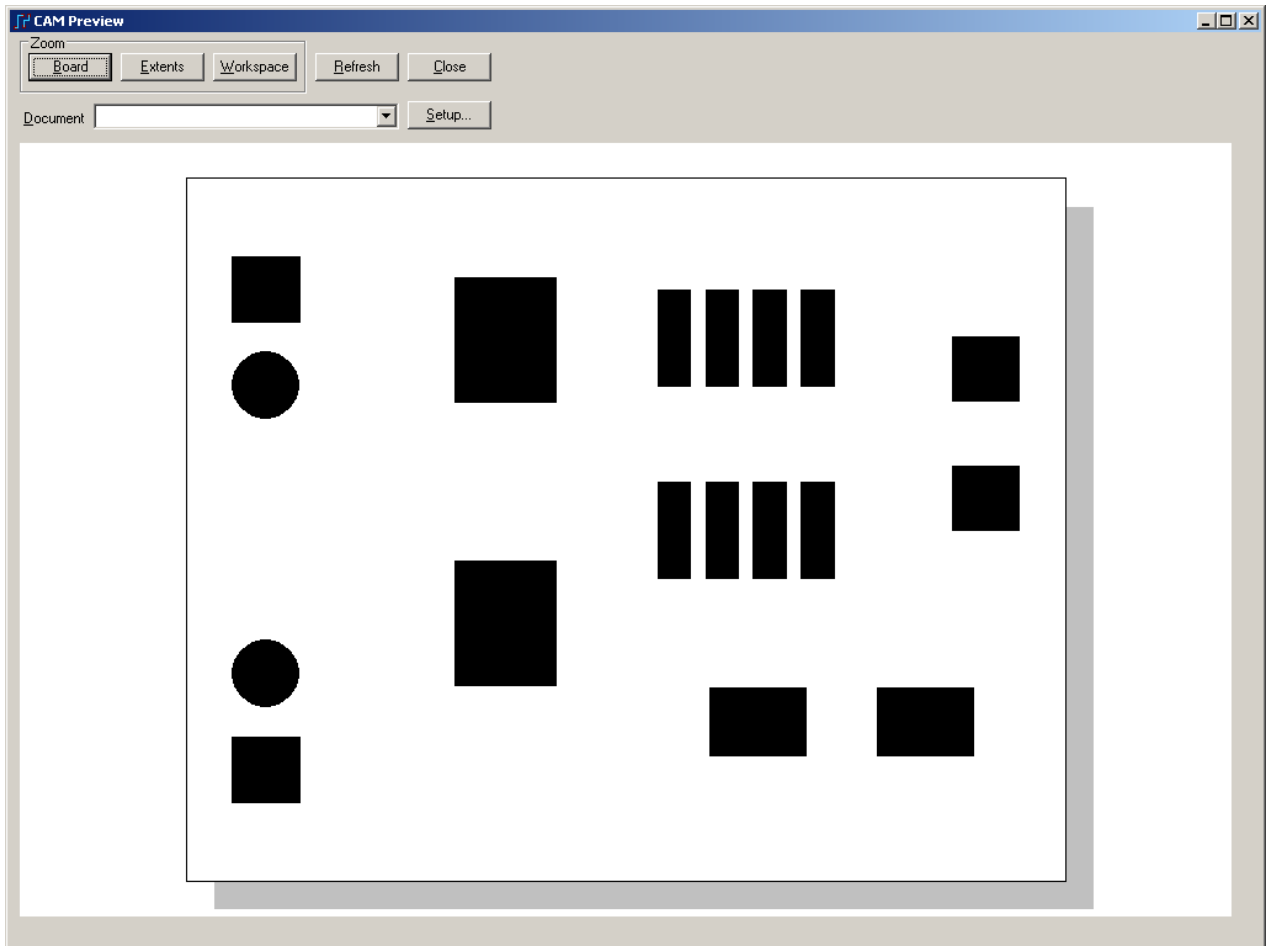
Press the **Add...** button.

Type **Solder Mask Top** for the Document Name.

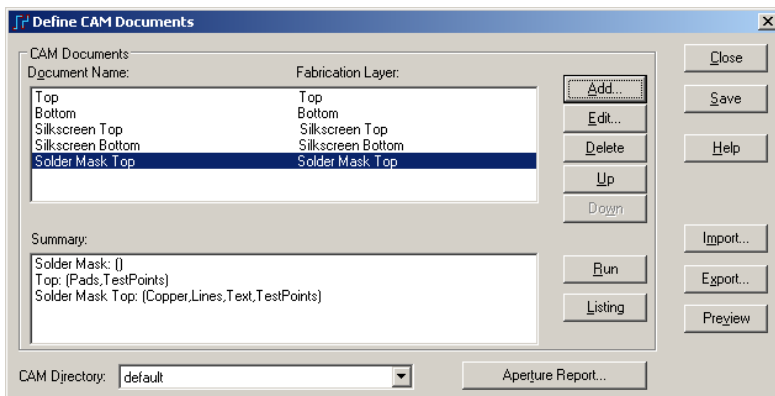
Select **Solder Mask** from the Document Type dropdown list and then select **Top** from the popup list.

Press the **Options** button and set the Justification to **Scale to Fit**. Press OK.

Press the **Preview Selections** button to verify your design.

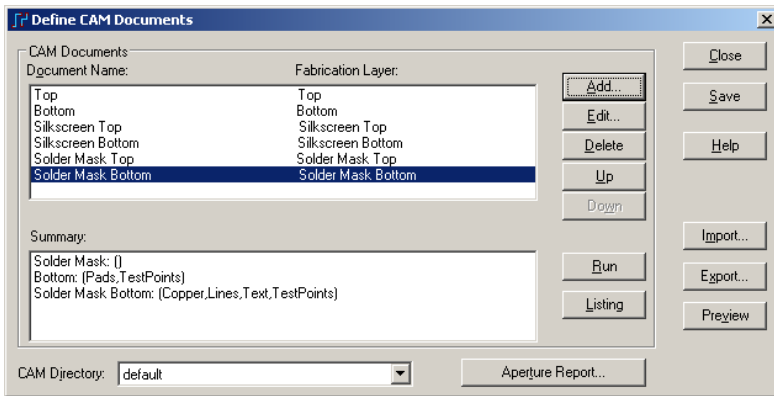


Close the preview and press OK to add the Solder Mask Top document.



5.7 Solder Mask Bottom.

Repeat Section 5.6, substituting Solder Mask Bottom for Solder Mask Top.



5.8 Drill Drawing

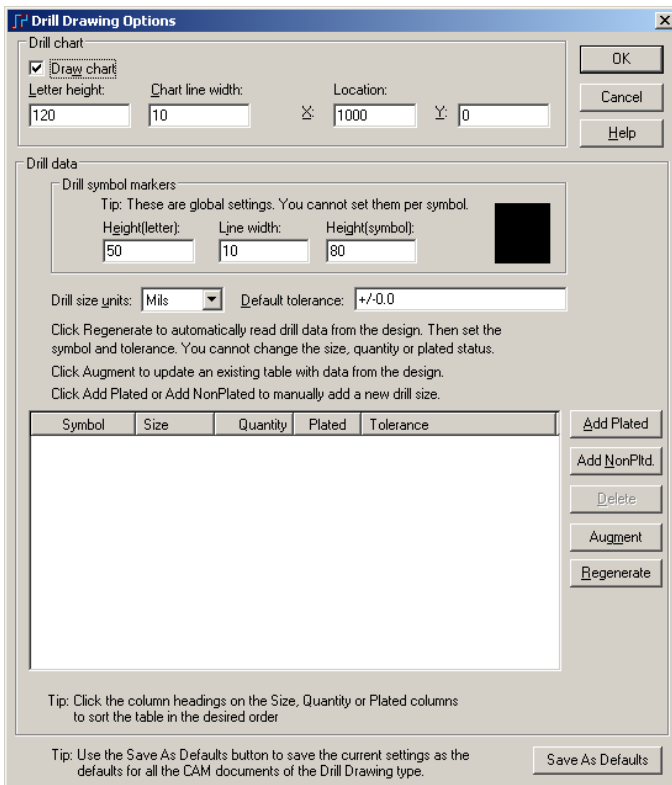
Press the **Add...** button.

Type **Drill Drawing** for the Document Name.

Select **Drill Drawing** from the Document Type dropdown list and then select **Top** from the popup list.

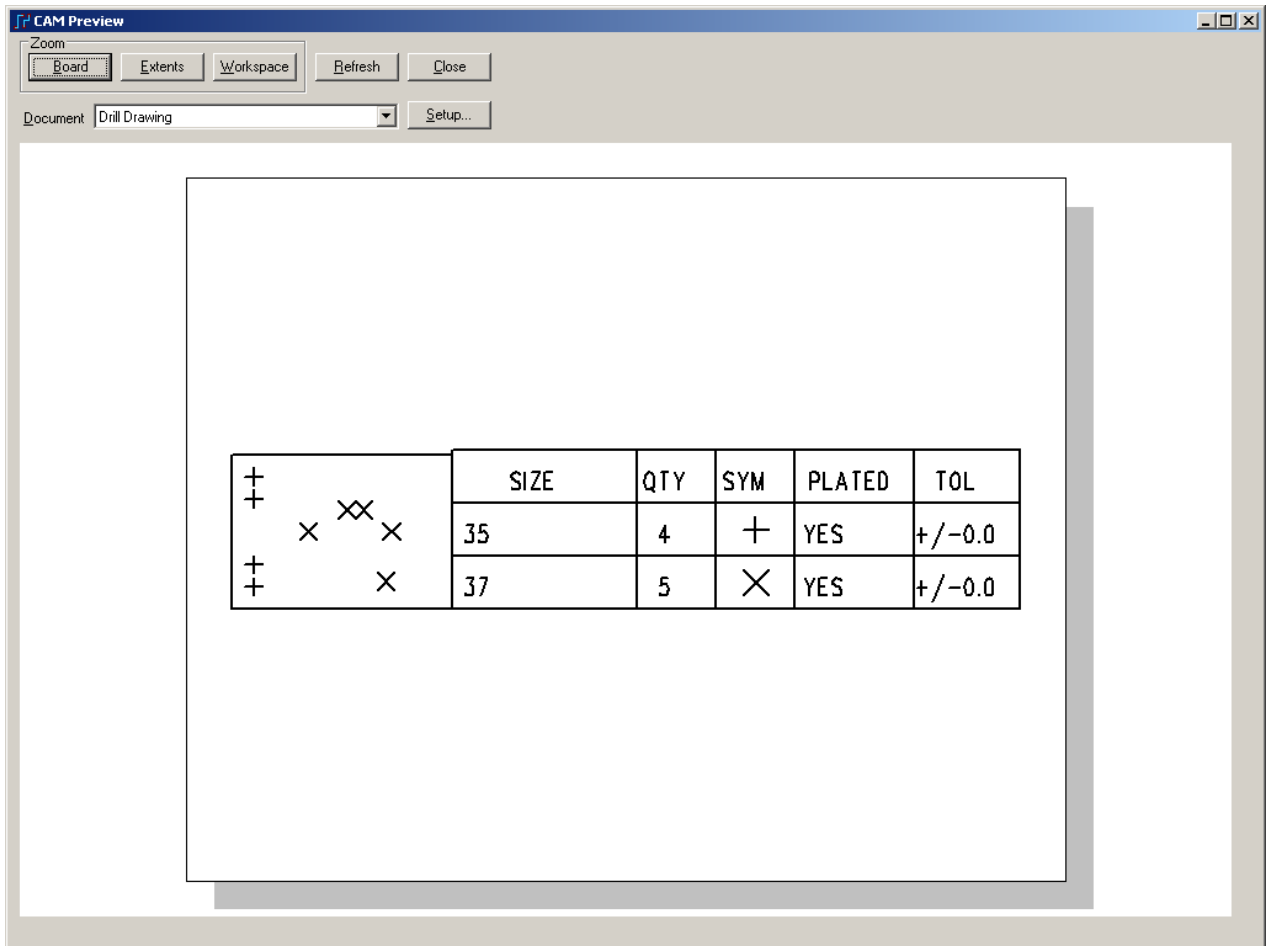
Press the **Options** button and set the Justification to **Scale to Fit**.

Press the **Drill Symbols** button and enter **1000** for Location X.

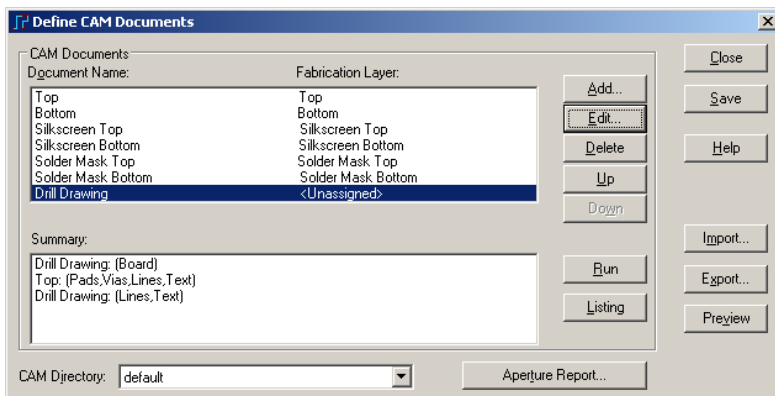


Press OK to close the Drill Drawing Options and Press OK again to return to the Add Document window.

Press the **Preview Selections** button to verify your design.



Close the preview and press OK to add the Drill Drawing document.



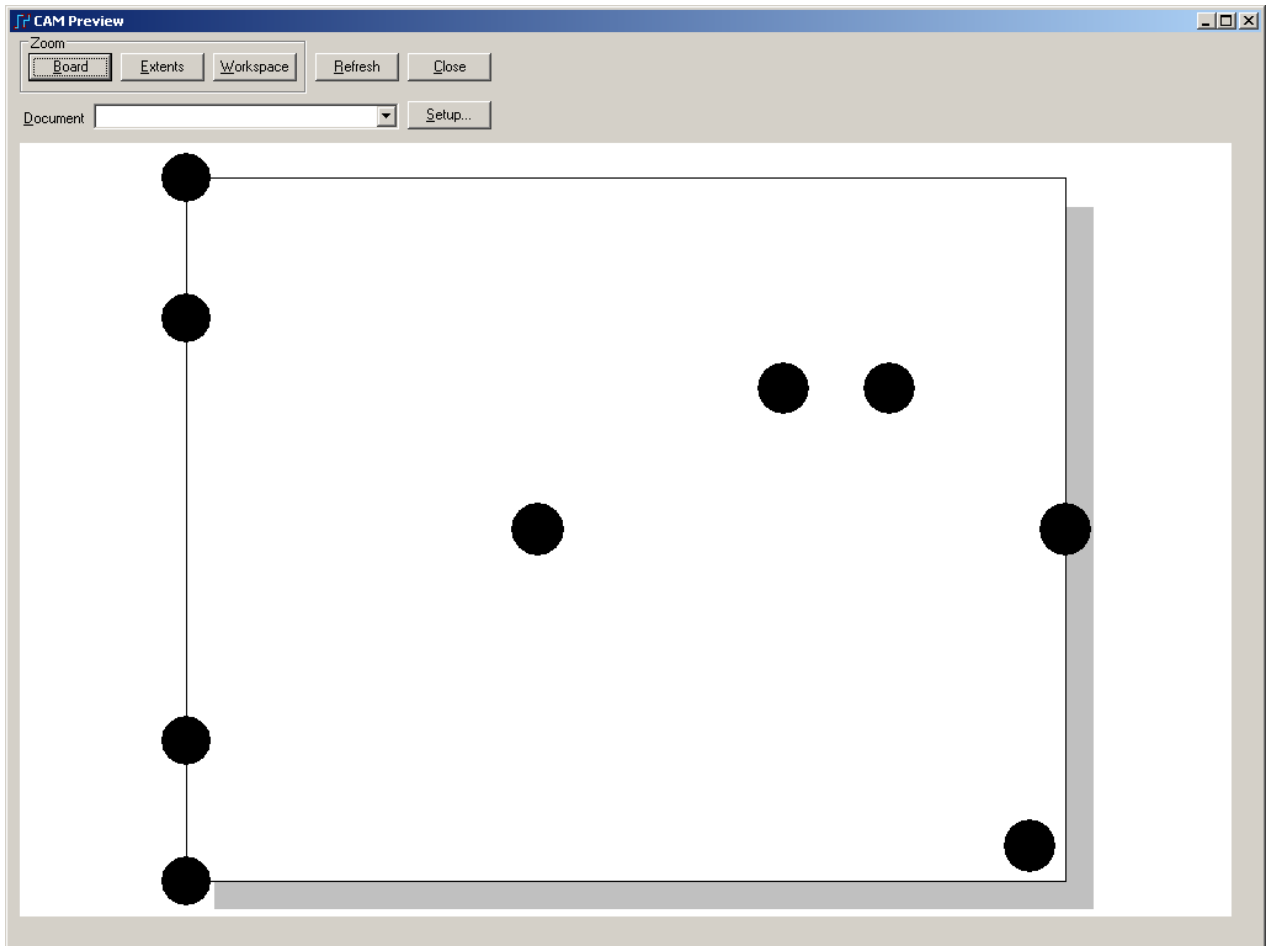
5.9 NC Drill Document

Press the **Add...** button.

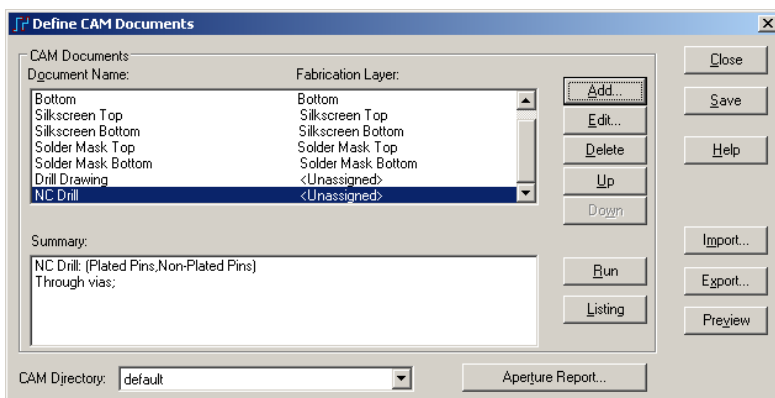
Type **NC Drill** for the Document Name.

Select **NC Drill** from the Document Type dropdown list.

Press the **Preview Selections** button to verify your design.

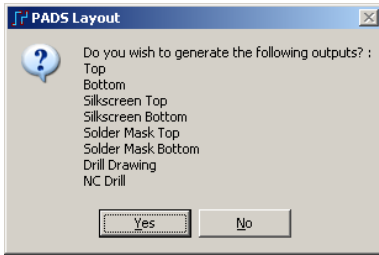


Close the preview and press OK to add the NC Drill document.



5.10 Generate the Gerber Files

Click on the **CAM Directory** dropdown and select **<Create>**. Enter **H:\PADS Tutorial\Cam** for the CAM Subdirectory. Press OK. Highlight all of the documents and press Run.



Click Yes to generate the Gerber Files.

After the files have been successfully generated, press Save and then close the CAM Documents window.

5.11 Verify Gerber Files

Before sending your Gerber Files to the PCB Company, it is recommended that you verify them with an external tool.

PCBExpress recommends using the PentaLogix ViewMate Gerber Viewer.

<http://www.pentalogix.com/Download/download.html>

You have now completed the Mentor Graphics PADS Tutorial.