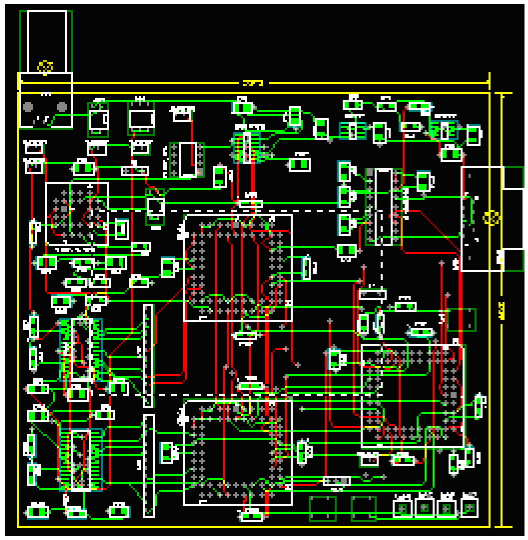
Orcad Layout Plus Tutorial



Layout Plus is a **circuit board layout tool** that accepts a layout-compatible circuit netlist (ex. from Capture CIS) and generates an output layout files that suitable for PCB fabrication. This tutorial is the second part of PCB project tutorial. Before start with Layout Plus, you need to have a complete netlist of your design, if you do not have it yet, please read the first part on "Capture CIS Tutorial".

Simple steps in producing PCB layout involve importing netlist, placing components, routing and generating output files and reports. For more information about Layout Plus, please refer to Layout Help (From Layout Plus menu, go to Help → Layout Help)

Opening Layout Plus and Creating New Design

To open Layout Plus, from Windows Start Menu, select **Program → Cadence PSD 15.0 → Layout Plus**. Go to **File → New** to create a new design. You will see the dialog as shown in figure 1. Enter the default technology template located on **C:\Cadence\PSD_15.0\tools\layout_plus\data_default.tch** in "Input Layout TCH" textbox. Enter the netlist (generated from Capture CIS) of your design in "Input MNL" textbox. And then enter the location and file name that you want the design file to be saved in "Output Layout" textbox. (You might want to save it on your network drive if you're running Capture CIS in campus's laboratory). Layout Plus will give the output layout file name as same as the input netlist file by default. If you change the output file, do not change the output file extension (.max). Then click Apply ECO.

Layout Plus will display the dialog shown in figure 2 after you apply ECO. In case of footprint linking error (missing footprints on some components), you will need to give the footprints to all missing components to complete AutoECO. (As explained in Capture CIS tutorial, it is better to specify footprints to all parts during schematic creation.) Hit "Accept this ECO" to import the netlist into Layout Plus. Figure3 shows the Layout Plus window after importing a netlist.

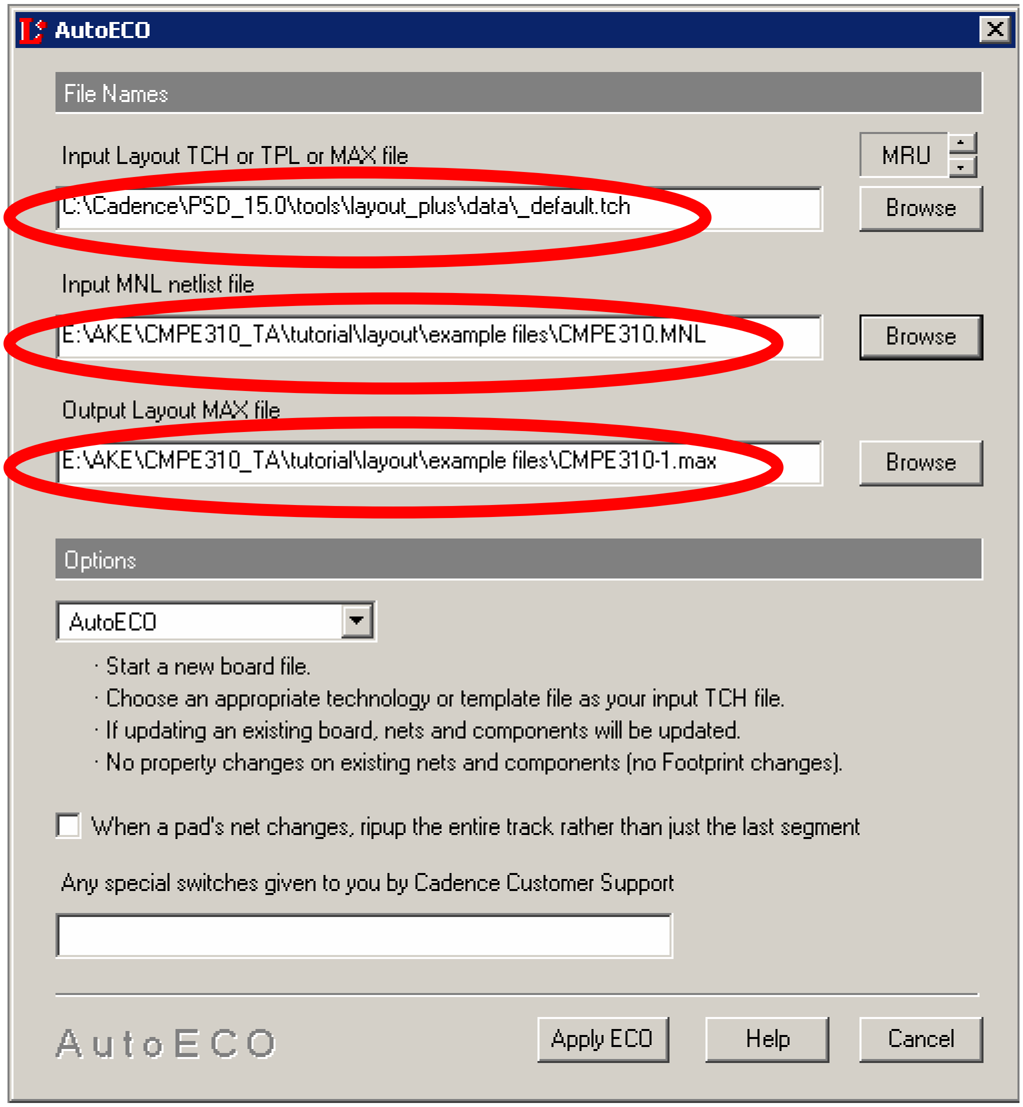


Figure 1: New Design Dialog

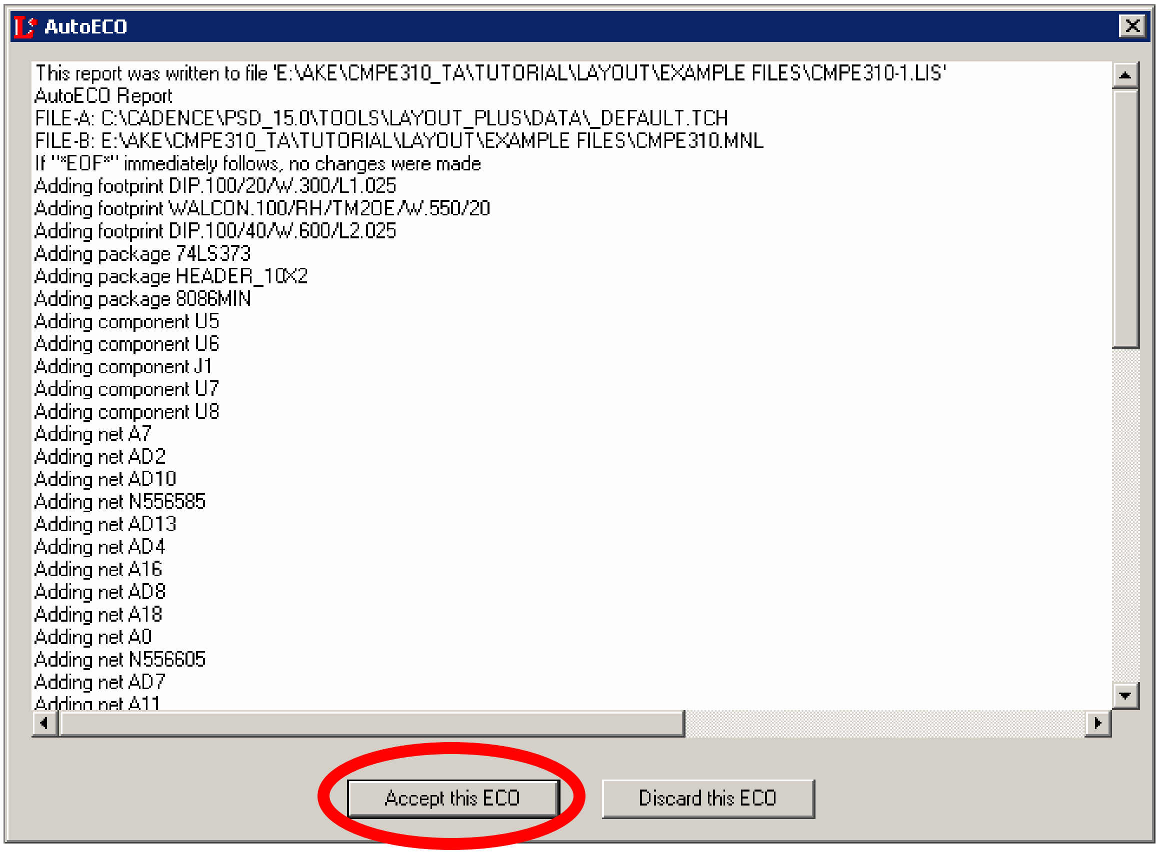


Figure 2: Dialog after applied ECO

More info:

- **ECO (Engineering Change Order)** is Layout Plus command that provide an ability to forward and back annotate your design flow. The forward annotate let you forward the change of your netlist (ex., from Capture CIS schematic) to the PCB. The back annotate just do the opposite, export the change of your design in PCB back to the schematic.
- You can set different options for AutoECO, the tool will update particular properties of the design according to the setting.
- Technology template (we are using _default.tch in this tutorial) contains information regarding layout design such as board layers, spacing, track widths, design rules, etc. Creating your own template is helpful in making a number of designs with the same set of rules and settings however it is out of this tutorial's scope.

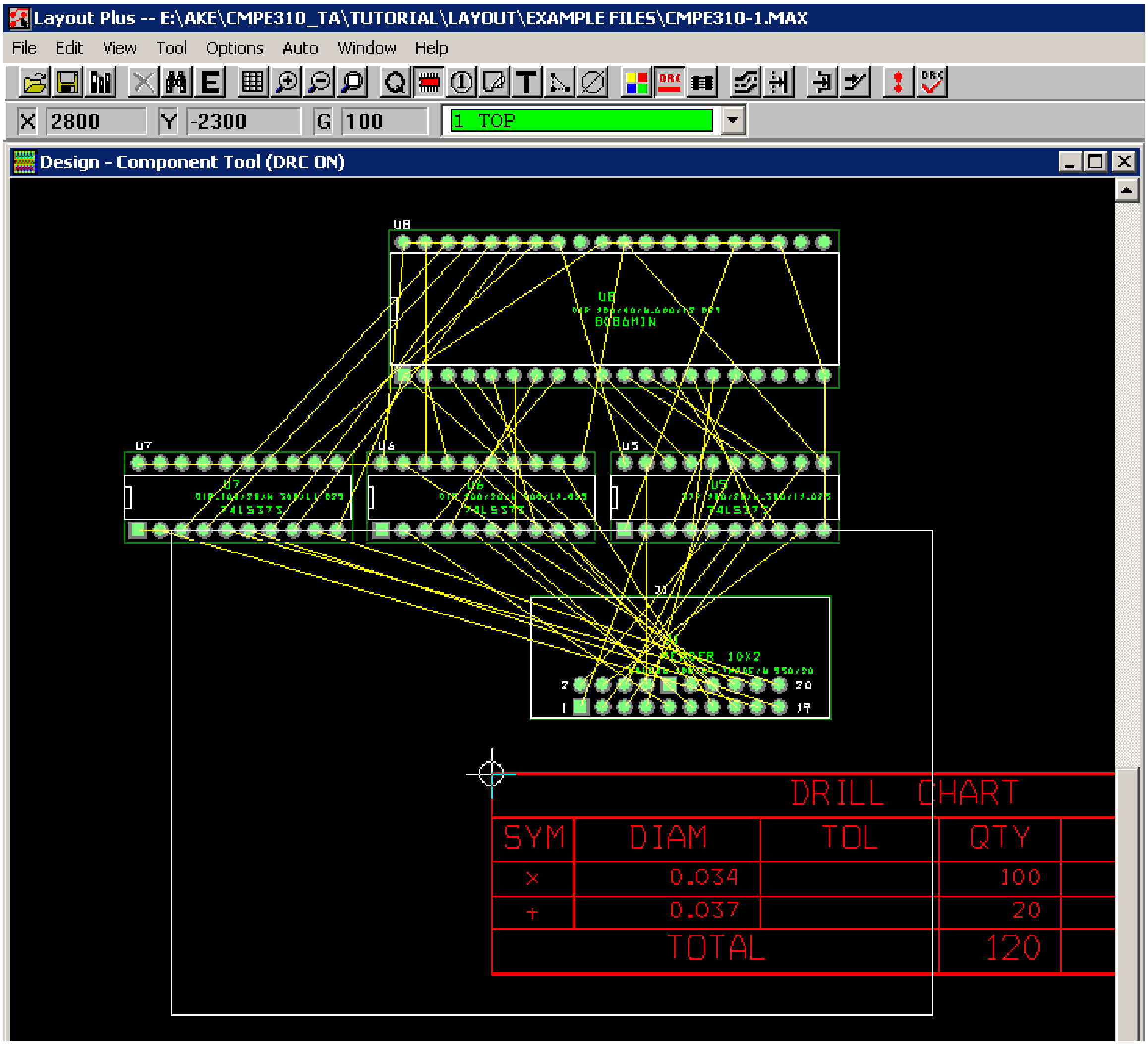


Figure 3: Layout Plus Window after importing netlist

Setting Design Environment

To set the design environment (display unit, grids, rotation, snap) go to **Options → System Settings**. Modify the setting that suitable for your design. Figure 4 shows the system setting dialog.

Next, you will need to define the layer stack for your design, go to **View → Database Spreadsheets → Layers**. From here, specify layer type and mirror layer (usually the most outer layers, TOP and BOTTOM, are the mirror layers of each other). An example of layer stack for 4-layer board design (TOP, BOTTOM, POWER and GND) with POWER and GND layers as power and ground plans and TOP and BOTTOM as routing layers is shown in figure 5.

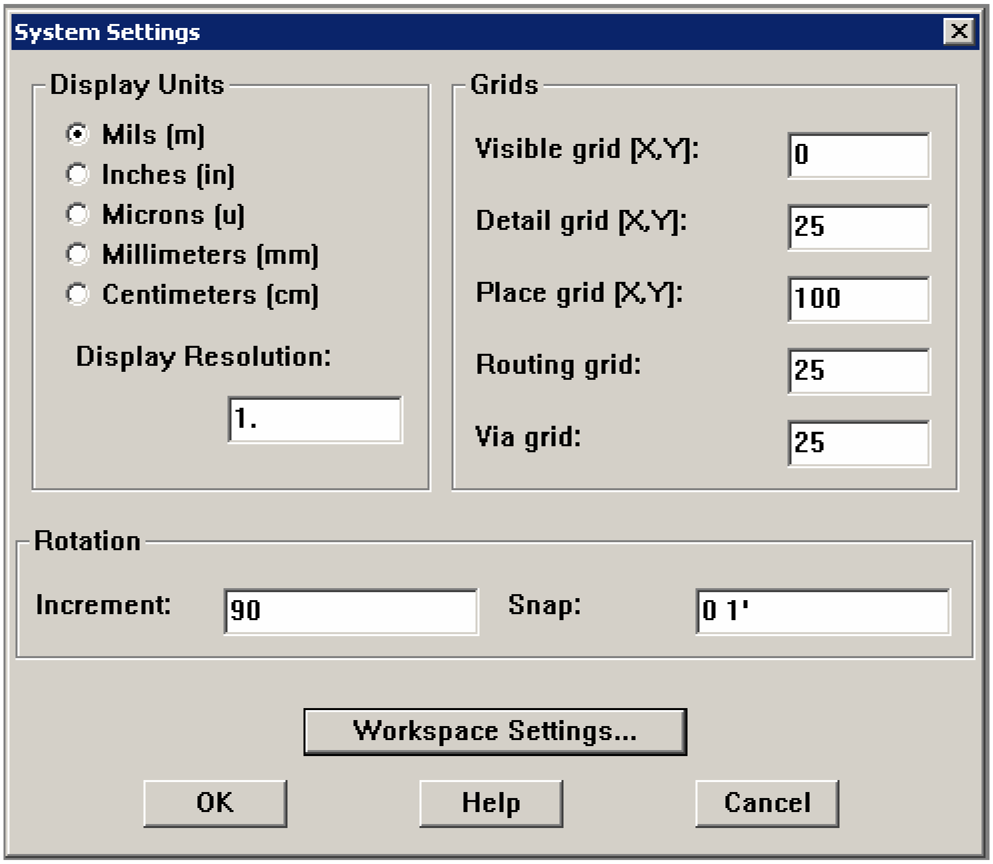


Figure 4: System Settings

After modify layer stack, you will need to specify routing spacing (**Options → Global Spacing**), you can modify track-to-track, track-to-via, track-to-pad, via-to-via, via-to-pad, and pad-to-pad spacing according to the capabilities of preferred PCB manufacturer. (for example, www.pcbexpress.com). Figure 6 shows all routing spacing set to 6 mil.

There are many other parameters that you can set and should be carefully checked with the recommended parameters from PCB manufacturer (drill sizes, padstacks, minimum track width, etc.) In this tutorial we will use all default parameter values. If you plan to fabricate your boards, please consult the PCB manufacturer.

More info:

- **Mil** unit is commonly used in PCB footprint and PCB board design.
- **1 mill = 0.001 inch**

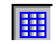
Layer Name	Layer Hotkey	Layer NickName	Layer Type	Mirror Layer
TOP	1	TOP	Routing	BOTTOM
BOTTOM	2	BOT	Routing	TOP
GND	3	GND	Plane	(None)
POWER	4	PWR	Plane	(None)
INNER1	5	IN1	Unused	(None)
INNER2	6	IN2	Unused	(None)
INNER3	7	IN3	Unused	(None)
INNER4	8	IN4	Unused	(None)
INNER5	9	IN5	Unused	(None)
INNER6	Ctrl + 0	IN6	Unused	(None)
INNER7	Ctrl + 1	IN7	Unused	(None)
INNER8	Ctrl + 2	IN8	Unused	(None)
INNER9	Ctrl + 3	IN9	Unused	(None)
INNER10	Ctrl + 4	I10	Unused	(None)
INNER11	Ctrl + 5	I11	Unused	(None)
INNER12	Ctrl + 6	I12	Unused	(None)
SMTOP	Ctrl + 7	SMT	Doc	SMBOT
SMBOT	Ctrl + 8	SMB	Doc	SMTOP
SPTOP	Ctrl + 9	SPT	Doc	SPBOT
SPBOT	Shift + 0	SPB	Doc	SPTOP
SSTOP	Shift + 1	SST	Doc	SSBOT
SSBOT	Shift + 2	SSB	Doc	SSTOP
ASYTOP	Shift + 3	AST	Doc	ASYBOT
ASYBOT	Shift + 4	ASB	Doc	ASYTOP
DRLDWG	Shift + 5	DRD	Doc	(None)
DRILL	Shift + 6	DRL	Drill	(None)
FABDWG	Shift + 7	FAB	Doc	(None)
NOTES	Shift + 8	NOT	Doc	(None)

Figure 5: 4-layer board with power and ground planes setting

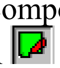
Layer Name	Track to Track	Track to Via	Track to Pad	Via to Via	Via to Pad	Pad to Pad
TOP	6	6	6	6	6	6
BOTTOM	6	6	6	6	6	6
GND	6	6	6	6	6	6
POWER	6	6	6	6	6	6
INNER1	6	6	6	6	6	6
INNER2	6	6	6	6	6	6
INNER3	6	6	6	6	6	6
INNER4	6	6	6	6	6	6
INNER5	6	6	6	6	6	6
INNER6	6	6	6	6	6	6
INNER7	6	6	6	6	6	6
INNER8	6	6	6	6	6	6
INNER9	6	6	6	6	6	6
INNER10	6	6	6	6	6	6
INNER11	6	6	6	6	6	6
INNER12	6	6	6	6	6	6
DRILL	6	6	6	6	6	6

Figure 6: Routing Spacing

Tips:

- Easy access to various designs setting by clicking on  button on the toolbar.
- Easier to work on your design by **Disable Online DRC** by clicking on  button on the toolbar

Creating Board Outline

Board Outline is the outline for all components and net routings of your PCB. To create board outline, click on Obstacle Tool  on the toolbar or go to **Tool → Obstacle → Select Tool**. Then double click on one corner point you want to make an outline, the Edit Obstacle dialog will popup, set obstacle type to "Board Outline" and obstacle layer to "Global Layer" as shown in figure 7.

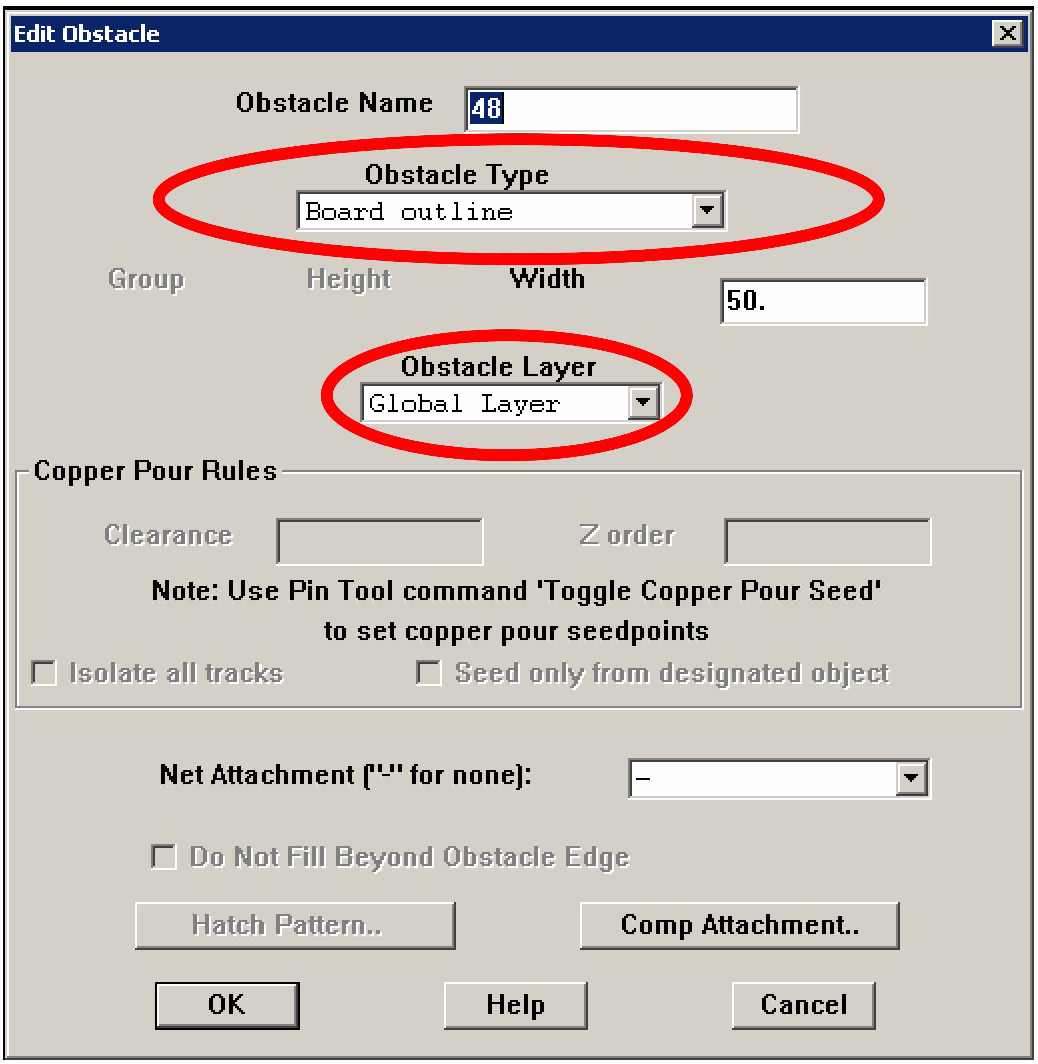


Figure 7: Edit Obstacle for Board Outline

Click OK and then draw your board outline as shown in figure 8. You will need to click on four corners of the obstacle, and then press ESC or left click and select "End Command".

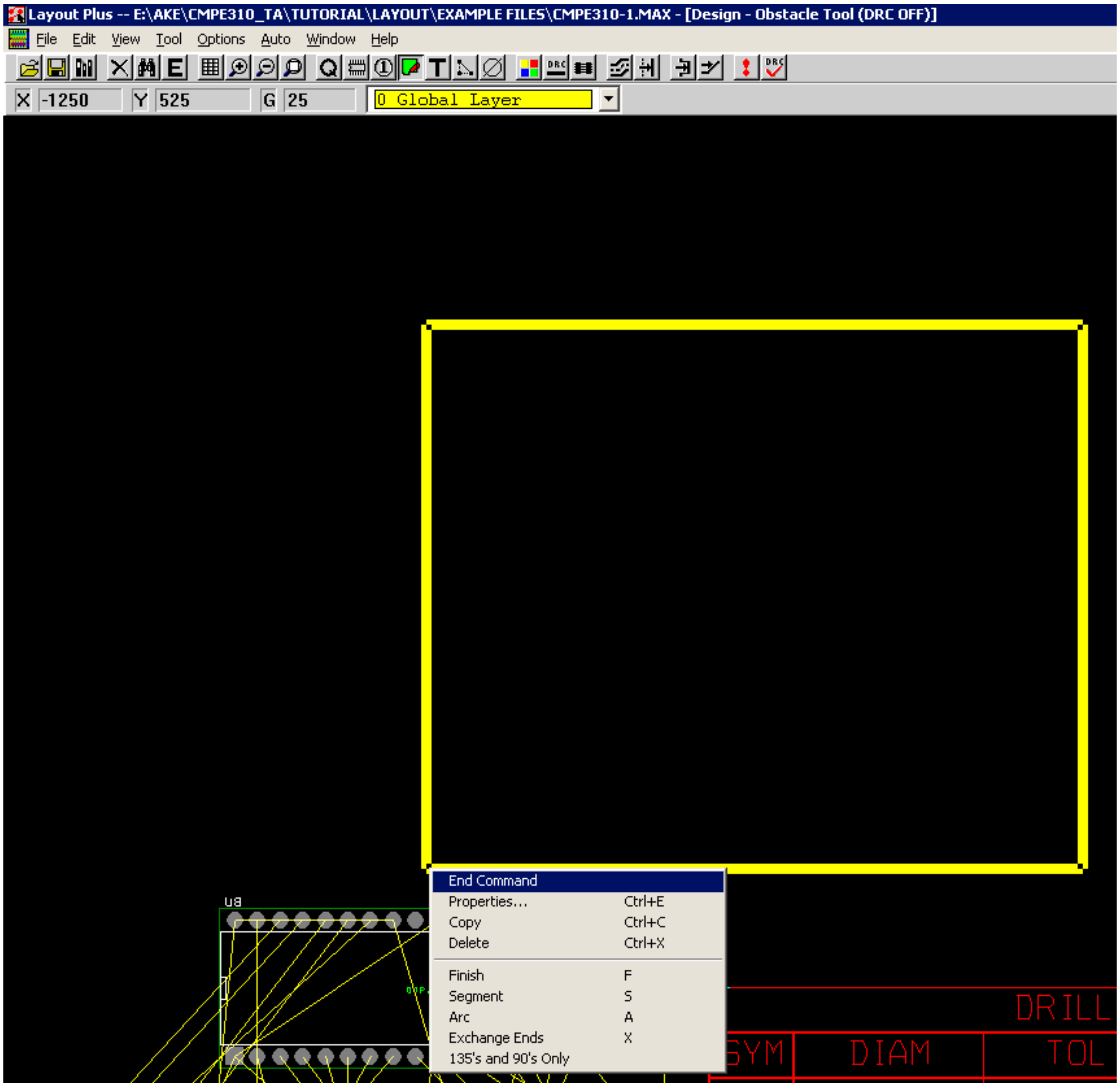
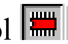


Figure 8: Board Outline Obstacle

Tips:

- Press **"I" for zoom-in and "O" for zoom-out**, the center of zooming is the position of the mouse. These shortcuts make it's very easy to work on your design.
- The size of board outline is an estimation of your board size, you can stretch it after placing components.

Placing components

You can either manually place components on the board or use auto placement feature of Layout Plus. To manually place components, click on Component Tool  on the toolbar or go to **Tool → Component → Select Tool**. Place all components inside the board outline.

For auto placement, first, you will need to **preplace** some of the components that you want them to be placed on specific locations (ex. power connectors and headers). Move the component to the desired location and then right click and select "**Lock**" or press **L**. Figure 9 shows the preplaced and locked header before auto placement.

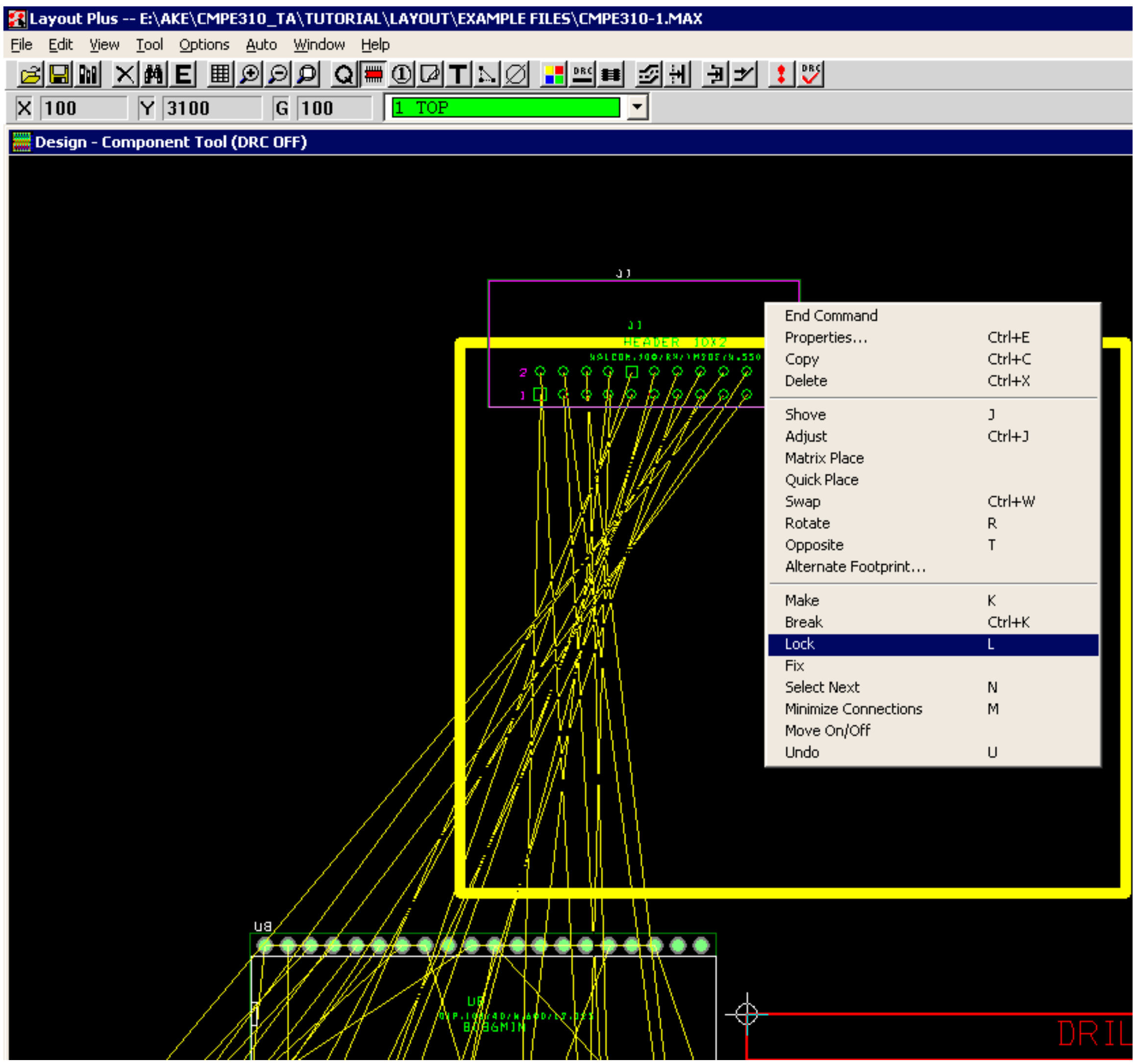


Figure 9: Preplace components

After preplace components, go to Auto → Place → Board. Layout plus will auto place components within your board outline. Then you can adjust the board outline to a proper size. Figure 10 shows the board after autoplaced.

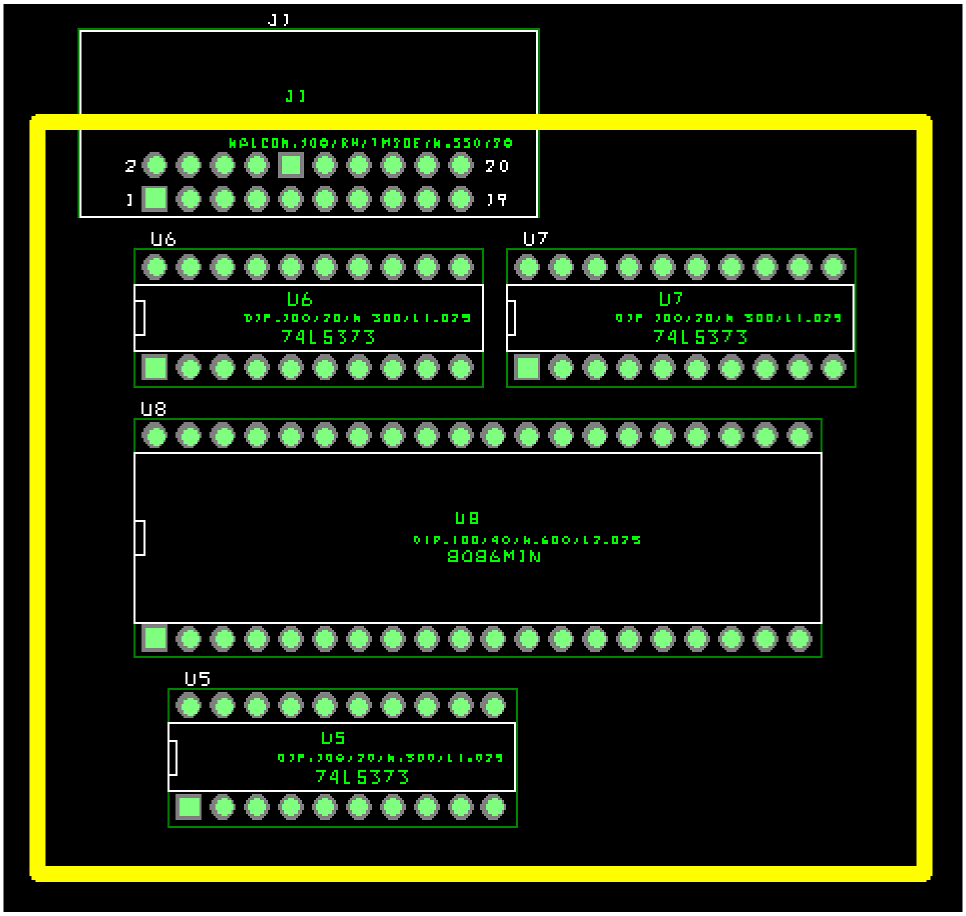


Figure 10: Board after autoplaced

Tips:

- For a large PCB design, you might want to let Layout Plus autoplace your components and then manually move some of the components to make your board compact.

Thermal Relief and Plane Layers/Copper zones

Thermal relief pad provides a connection from pin or fanout to power and ground plane layer or copper zone while minimize the heat transfer to plane during soldering. For example, if you have 4-layer boards with power and ground planes, there will be thermal relief pads connect power and ground pins to power and ground planes. You can set the size of thermal relief pad by go to **Options → Thermal Relief Setting**.

Routing the board

This section will show simple steps involved in autorouting. For manual routing, please refer to Layout Plus Help. Before start routing, you need to assign plane layers to power and ground net and disable routing for power and ground nets. First, go to **View → Database Spreadsheets → Nets**. The nets spreadsheet dialog will show up, double click on **power net (VCC), uncheck "Routing Enabled"** and then click on **Net Layers** button. From **Layers Enabled for Routing** dialog, select the layer that you set to be used as a power plane (ex. POWER) in "Plane Layers" box as shown in figure 11. Repeat the same step for ground net (GND) except assign the other layer for ground plane (ex. GND). After disable routing on power and ground nets and assign appropriate layers to them, **make sure that all other nets have a routing enabled attribute**.

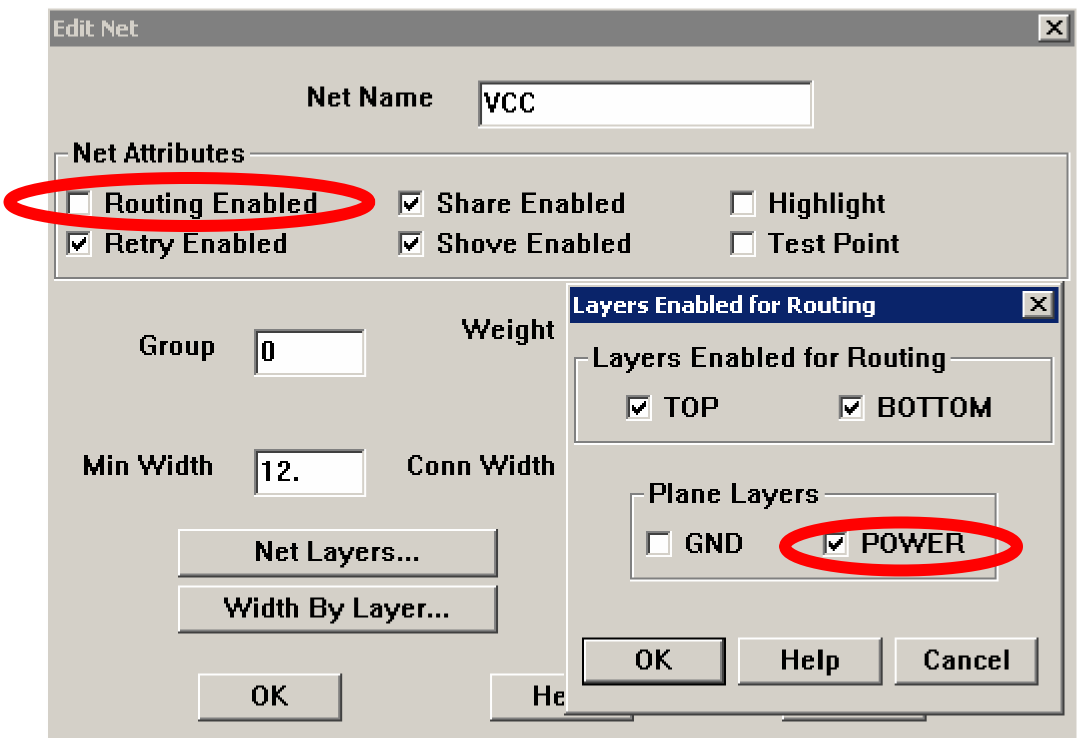


Figure 11: Disable routing and specify plane layer for power and ground net

For **SMT (Surface Mount Technology)** board, you need to create **fanout** to route a surface mount pad to via which provide a way to route from the pad layer to any other layers. To create fanout, go to **Auto → Fanout → Board**. Skip this step if you are working on through-hole board.

Next step is to perform an autoroute, go to **Auto → Autoroute → Board**. The result of this step is shown in figure 12.

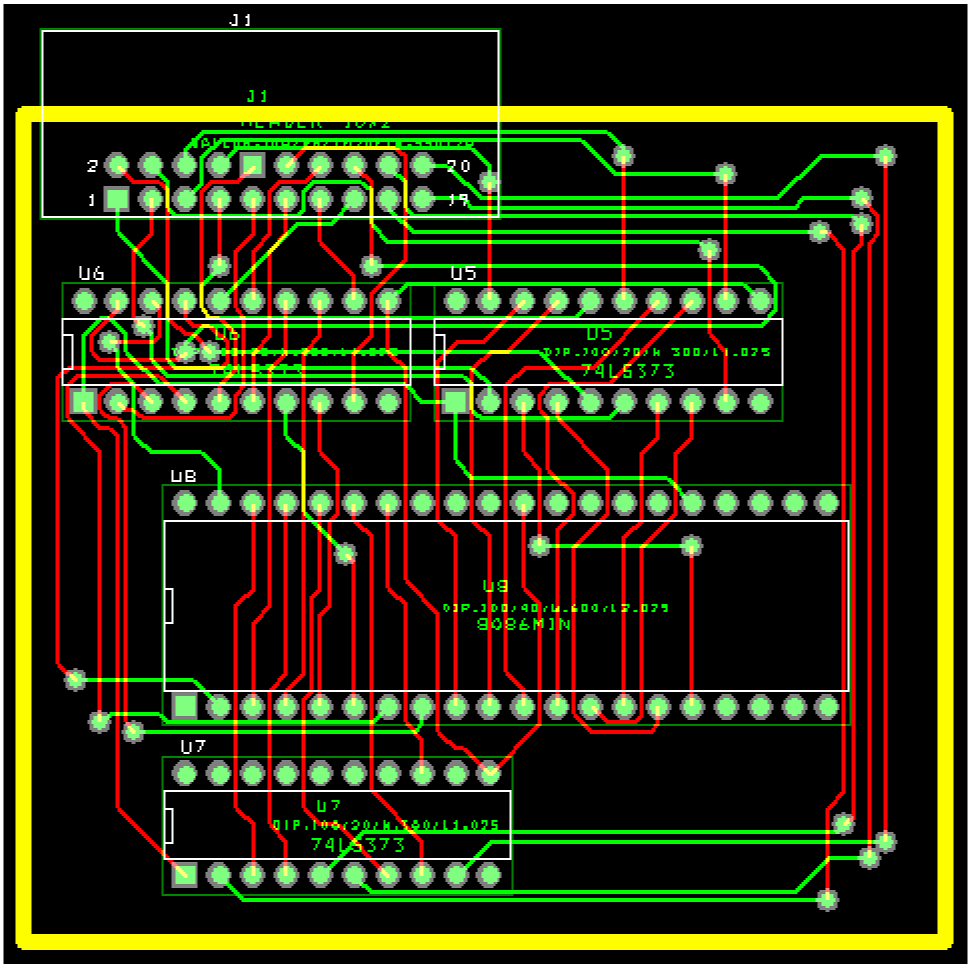


Figure 12: Board after autorouting

Tips:

- Your can check each layer connection by turn on each layer display at a time, first press **Backspace** to turn off all layers and then press key number associated with each layer (ex. **0** for Global layer, **1** for TOP layer, **2** for BOTTOM layer, **3** for GND layer, **4** for POWER layer and so on)
- To turn off one layer at a time, select the layer you want to turn off from the toolbar (or press key associated with the layer) and then press **– (minus sign)**
- The key assigned to each layer is specified in **Layer Spreadsheet**

Design Rules Check

To run DRC, go to **Auto → Design Rules Check**. You can select options you want to be checked, DRC command will use system environment parameters that you specified (or default values if you did not modify) as the design rules.

Print PCB layout

Go to File → Print/Plot, the dialog as shown in figure 13 will show up. You can check the option "**Print/Plot Current View**" if you want to print only the layer(s) that are being displayed.

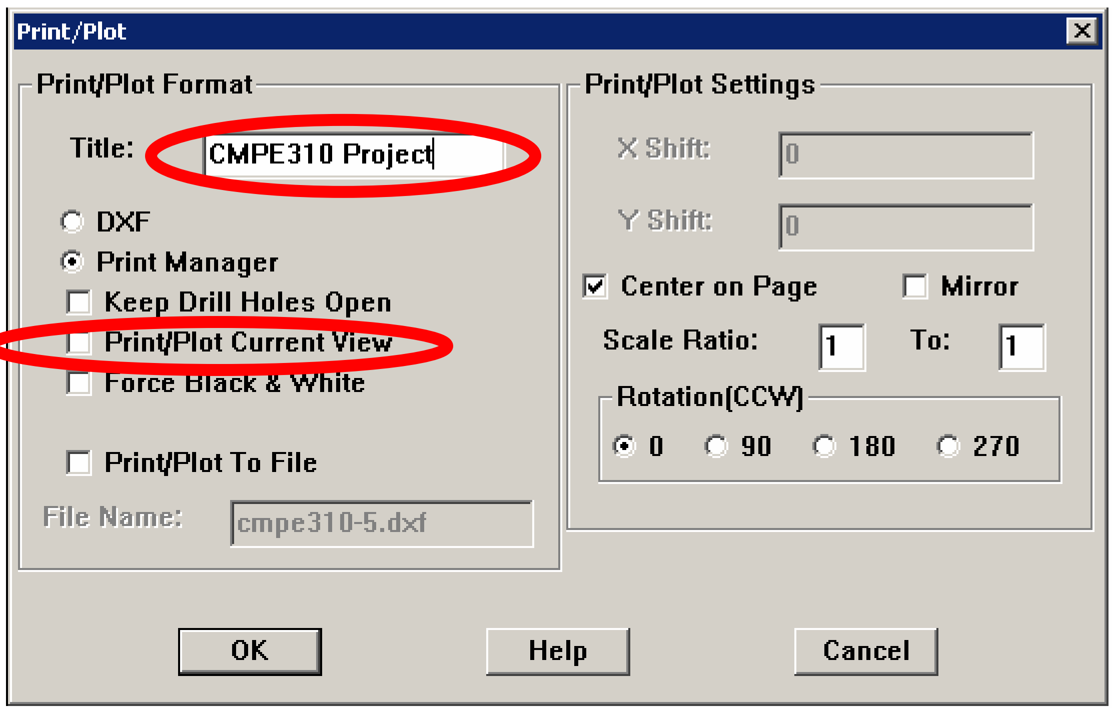


Figure 13: Print/Plot Dialog

Post Process

Now, you should have a completed PCB design (with no error from DRC), next you would want to generate the output files for PCB fabrication. The industry standard file format for PCB fabrication is called **Gerber files**. These gerber files tell the machines in PCB production process on how to draw patterns, make traces, drill holes and cut board.

First, you need to set the post processor by go to **Options → Post Process Settings**, from the dialog as shown in figure 14, you can choose the output format as GERBER RS-274D or Extended GERBER (RS-274X) (depends on PCB manufacturer requirement). After set the post processor, go to **Auto → Run Post Processor** to generate the output Gerber files.

Post Process

Plot output File Name	Batch Enabled	Device	Shift
*.TOP	Yes	EXTENDED GERBER	No shift
*.BOT	Yes	EXTENDED GERBER	No shift
*.GND	Yes	EXTENDED GERBER	No shift
*.PWR	Yes	EXTENDED GERBER	No shift
*.IN1	No	EXTENDED GERBER	No shift
*.IN2	No	EXTENDED GERBER	No shift
*.IN3	No	EXTENDED GERBER	No shift
*.IN4	No	EXTENDED GERBER	No shift
*.IN5	No	EXTENDED GERBER	No shift
*.IN6	No	EXTENDED GERBER	No shift
*.IN7	No	EXTENDED GERBER	No shift
*.IN8	No	EXTENDED GERBER	No shift
*.IN9	No	EXTENDED GERBER	No shift
*.I10	No	EXTENDED GERBER	No shift
*.I11	No	EXTENDED GERBER	No shift
*.I12	No	EXTENDED GERBER	No shift
*.SMT	Yes	EXTENDED GERBER	No shift
*.SMB	Yes	EXTENDED GERBER	No shift
*.SPT	No	EXTENDED GERBER	No shift
*.SPB	No	EXTENDED GERBER	No shift
*.SST	Yes	EXTENDED GERBER	No shift
*.SSB	No	EXTENDED GERBER	No shift
*.AST	Yes	EXTENDED GERBER	No shift
*.ASB	No	EXTENDED GERBER	No shift
*.DRD	Yes	EXTENDED GERBER	No shift

Figure 14: Post Process Setting