PADS Layout & Router Demo

Seneca College of Applied Arts and Technology
School of Information and Communication Technology

Introduction

Mentor Graphics is the leading printed circuit board (PCB) design software company. They produce many types of software to create and simulate circuits and to design PCBs. This document outlines one combination of Mentor Graphics software that they call the **PADS Flow**.

PADS Suite

The PADS Suite is a collection of programs with all the tools necessary to create a PCB. It is designed to be used by individuals and small teams but is still very powerful.

PADS contains three applications:

PADS Logic   for design entry, i.e. to create the circuit diagram

PADS Layout   to layout components on a PCB.

PADS Router   to create traces between the PCB components.

DxDesigner

DxDesigner is another design entry program. PADS Logic is designed to be used with Layout and Router but DxDesigner can be used with many other tools, including PADS and even those from competing companies. For example, DxDesigner is tightly integrated with Expedition Enterprise, an application similar to PADS Layout and Router but designed for collaboration between large teams.

The features present in DxDesigner, such as the ability to break a circuit down into small blocks of components, allows large and complicated circuits to be created more easily than in PADS Logic. It also contains a tool called DxDatabook which allows a user to search a database for components based on the required criteria.
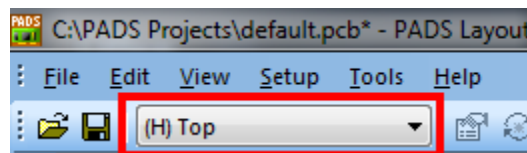
Getting Started

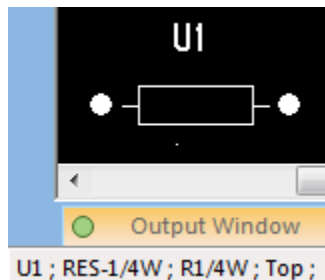Run PADS Layout by double-clicking the desktop icon.

Create a new file by clicking *File menu >New*. If there is a prompt to set the start-up file, select the *System Default Start-up File* and press OK.

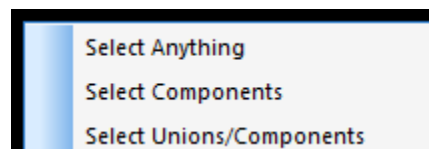There are three things to be aware of when using PADS:

1. The layer you are working on. The current layer is shown in the drop-down box in the toolbar. Components should generally go on the top layer and traces on the bottom.

2. The object that is selected. The selected object is shown in the status bar and is highlighted in white. If you change modes while something is selected, the mode will be directly applied to the selection. For example, when creating routes between parts you normally click where you want to start a trace then click where you want it to end. If an object is already selected, changing to routing mode will set the start of the trace as whatever is selected. This behaviour can be confusing so it is recommended to **make sure that nothing is selected when changing modes**.

3. The selection filter which determined what objects can be selected by clicking on them. With nothing selected, you can right-click the working area and choose one of the "Select …" options. If you cannot select something it means you need to change the filter.
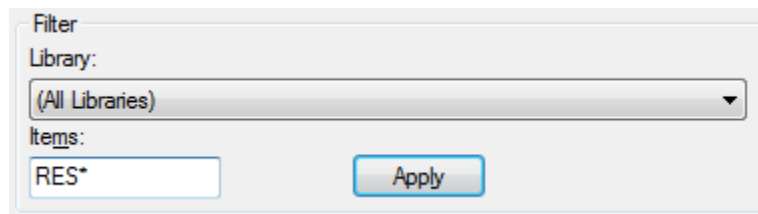
Adding Parts

Enable the ECO Toolbar by clicking the toolbar button. Press *OK* at the prompt.
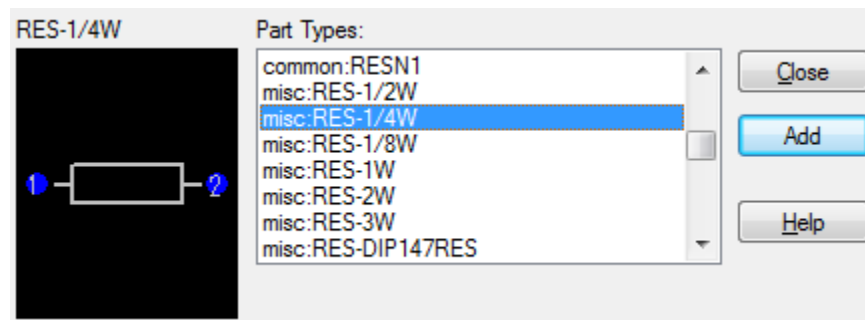
In the new toolbar that appears, press the *Add Component* button.

Enter "RES*" into the items textbox and press *Apply*. All parts that start with RES (i.e. a resistor) will be shown in the listbox.
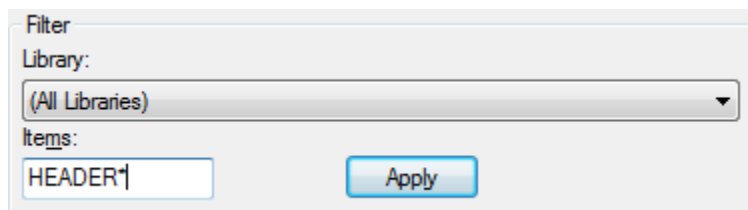
Find *misc:RES-1/4W* and select it. The name indicates that the part is in the *misc* library and is called *RES-1/4W* which means it is a standard ¼ watt through-hole resistor. The parts list will show various other resistors including surface mount types.

Press *Add* then click on a blank part of the working area to place the component.

Change the filter to "HEADER*" and press *Apply*.

Find *connect:HEADER02*, press *Add*, and click the working area to add the header. Press *Close*.

Click the resistor and click again in an empty spot to duplicate it. Do this again so you have three resistors. See the next page for an example.
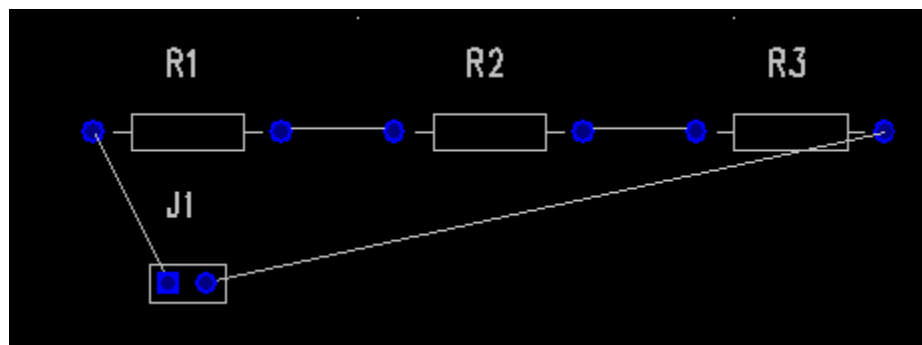
Connecting Components

Press the *Add Connection* button in the ECO Toolbar.

With nothing selected, click the right pin of a resistor then click the left pin of the resistor next to it and press ESC. Repeat this to connect the all objects in series as shown below.
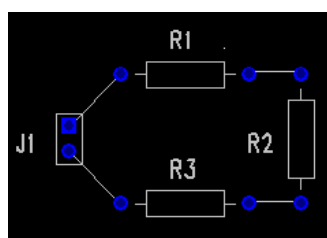

Layout

Switch to the drafting toolbar.

Objects are moved by first clicking and dragging to attach the object to the cursor then clicking to place it. You can also select an object and press CTRL+E to start moving it.

Objects are rotated by selecting the object then right clicking and selecting Rotate 90. You can also press CTRL+R.

Place and rotate the objects as shown below. You will need to move both components and reference designators.

Text

Click the *Add Text* button.

Enter your name as the text and change the layer to Silkscreen Top. Press *i* and click near your components to add the text.

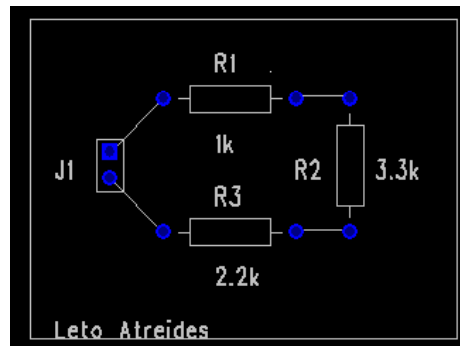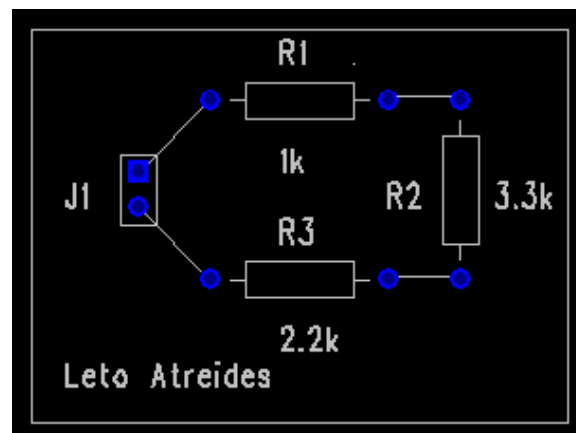Add more text to label the resistor values as shown below.

Board Outline

Press *the Board Outline and Copper Cutout* button.

Right click an empty area and select *Rectangle*. Click at the top-left of the components then click the bottom-right to create a rectangle around everything.
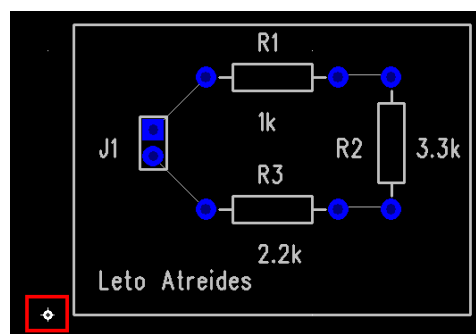

Right click an empty area and change the selection filter to Board Outline. Drag the edges of the outline close to the objects.

Origin

PADS uses an XY grid and the origin is the (0,0) point. Click *Setup menu > Set Origin* then click below and to the left of your design and press *Yes*.
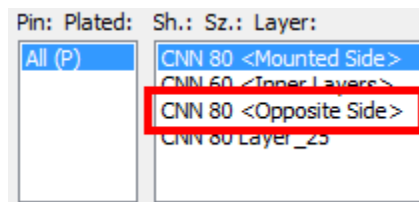
Pad Stacks

A "pad stack" is the set of drill holes and copper pads that a component uses. The default sizes from the library are designed for automated machine soldering not hand soldering. You will need to increase the pad size so that hand soldering is much easier.

Click *Setup menu > Pad Stacks*. You will need to change the diameter from 60 mils (1.5 mm) to 80 mils (2 mm) for each relevant decal, pin, and layer.
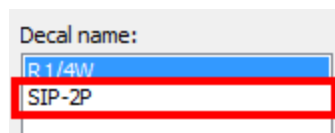
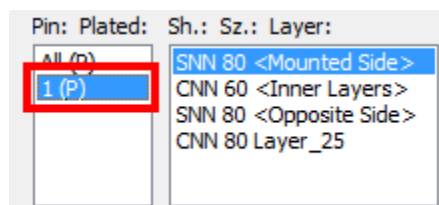The R1/4W decal and *Mounted Side* layer will selected already. Change the *Diameter* to 80.

Switch to the *Opposite Side* layer. Change the *Diameter* to 80.

Change the decal to SIP-2P and press *Yes* at the prompts. Change the diameter in both the mounted and opposite side layers.

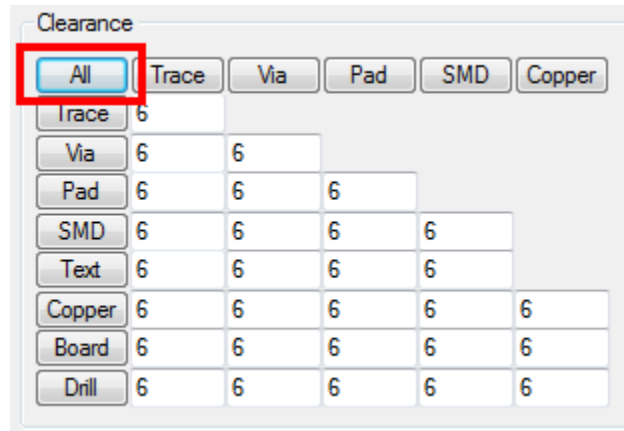Change to pin 1. Change the diameter in both the mounted and opposite side layers.

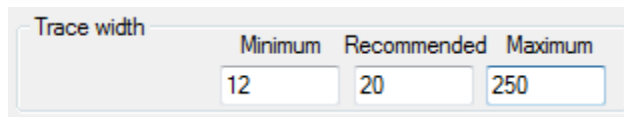

Press OK.

Design Rules

Design rules specify things such as the minimum space between components and the recommended trace size.

Press *Setup menu > Design Rules*. Click *Default* then *Clearance*.

Press the *All* clearance button, enter 12, and press OK.

Change the recommended trace width to 20 and the maximum to 250.

Press *OK* and close the two design rules windows.

Routing

Before continuing, **save your file**. If you do not, the program will crash in the following steps and you will lose your work.

Press the *Route* button. This will open PADS Router. It is possible to route within PADS Layout but Router is specifically designed for the task.
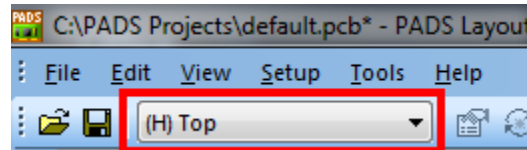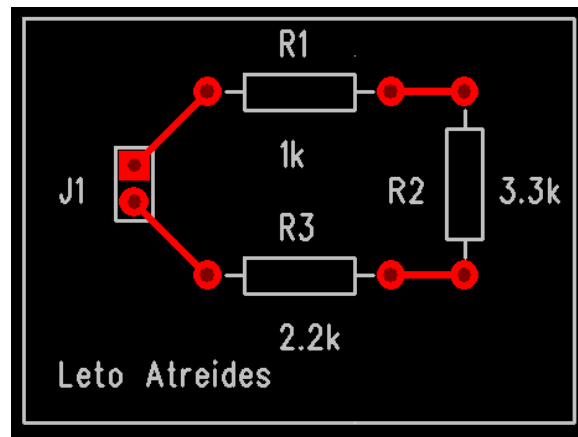

Enable the *Route Editing* toolbar.

With nothing selected, press the *Interactive Route* button.

Change the layer to *Bottom*.

Click the top pin of J1 then click the left pin of R1. This will create a trace for that connection. Do this for the other three connections. Your design will appear as shown below.
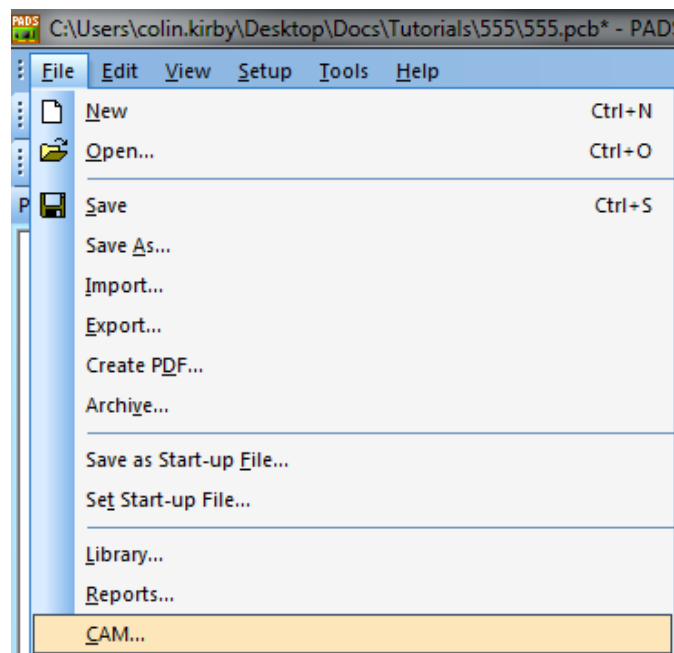

Exporting CAM Documents

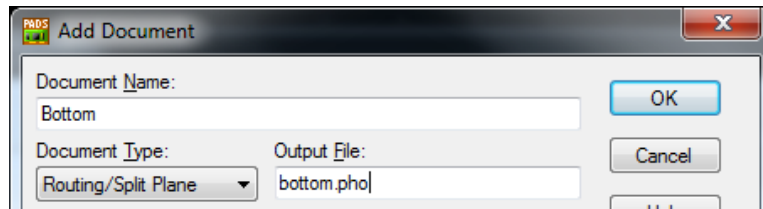Press the *Layout* button to return to PADS Layout.

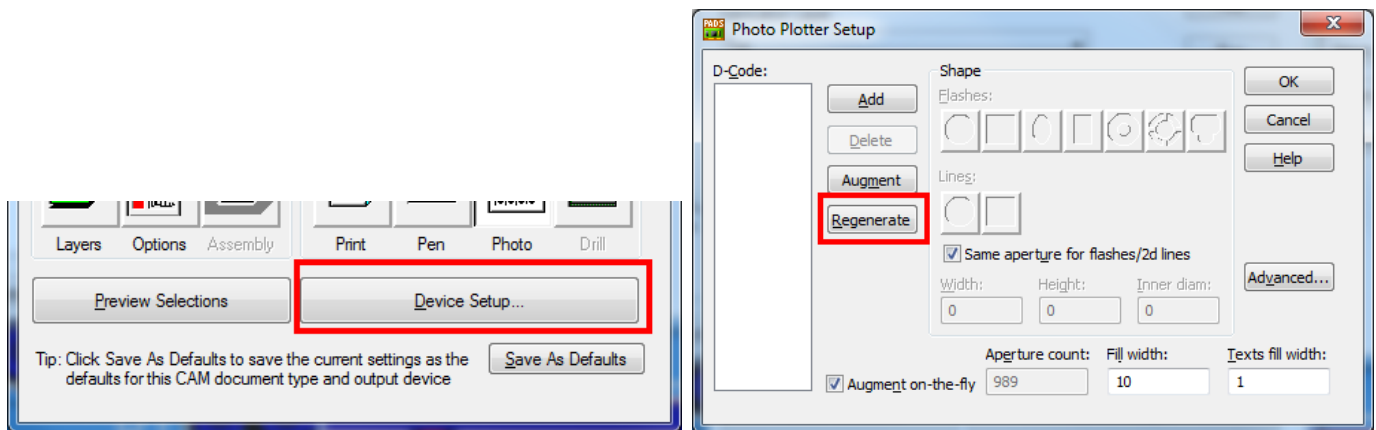Select *File menu > CAM* to open the *Define CAM Documents* window.

Click the *Add* button. Change the *Document Name* to "Bottom".

Change the *Document Type* to *Routing/Split Plane* and select the *Bottom* layer.

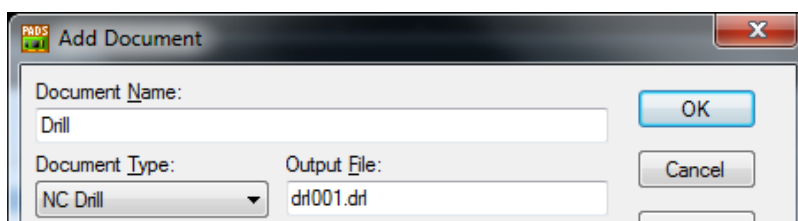Change the *Output File* to "bottom.pho".

Press *Device Setup* then *Regenerate*. Click *OK* and *OK* again to return to the documents window.

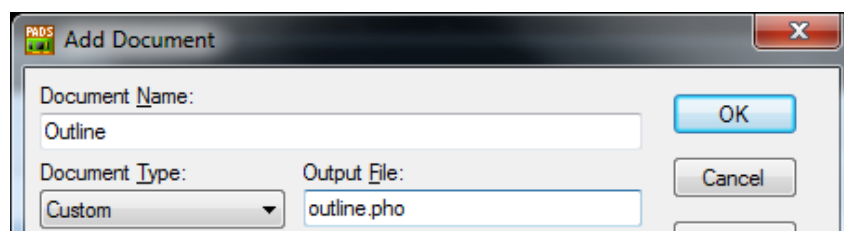Click the *Add* button. Change the *Document Name* to "Drill".
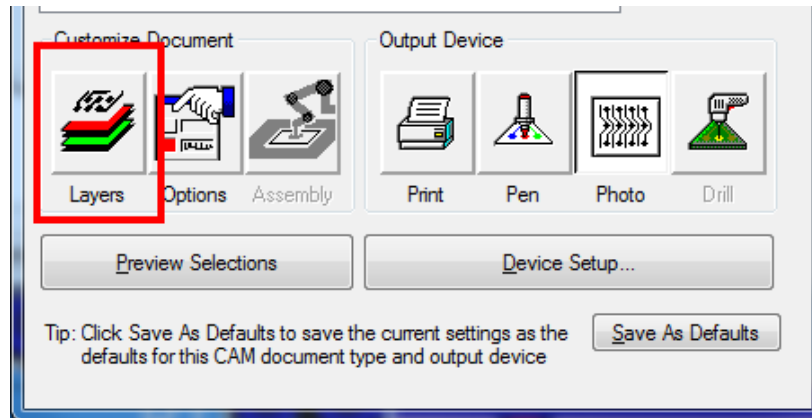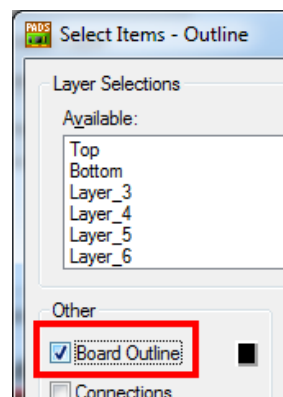
Change the *Document Type* to *NC Drill*.

Press *OK.*

Click the *Add* button. Change the *Document Name* to "Outline".

Change the *Output File* to "outline.pho".

Press the *Layers* button.

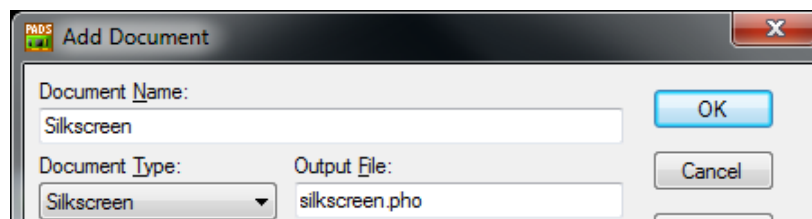Select *Board Outline* and press *OK* and *OK* again.

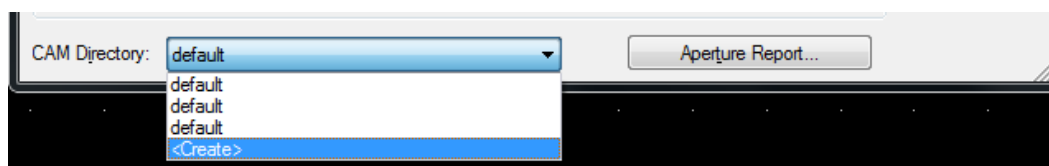Click the *Add* button. Change the *Document Name* to "Silkscreen".

Change the *Document Type* to *Silkscreen* and select the *Top* layer.

Change the *Output File* to "silkscreen.pho".
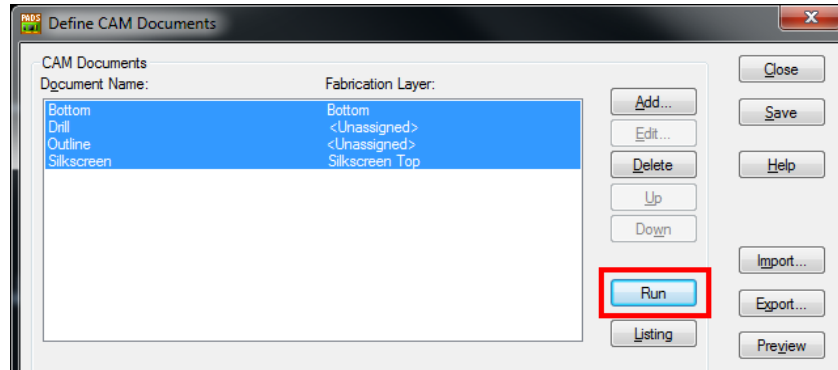
Press *OK*.

Press the *CAM Directory* dropdown box and select *Create*.

Press *Browse*. Locate where you saved your design and press *OK*. Press *OK* again.

Click and drag across all the documents in the window to select them all. Now click *Run* and *Yes* to create the files.

Press *Close* and *Yes* if it asks to save changes.