

Using DesignWorks Professional for the Mac with Osmond PCB

January 14, 2008

This technical note outlines how to set up designs in DesignWorks Professional for easy transfer to Osmond/PCB.

Topics To Be Added

- Creating Power and Ground Connections
- Multigate packaging

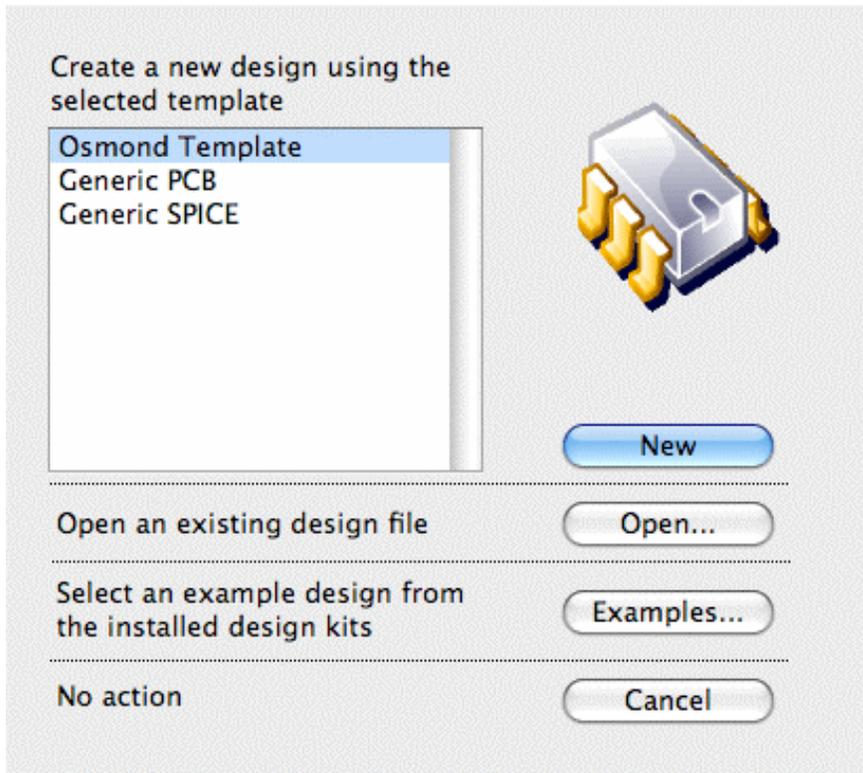
Legal Notes

DesignWorks Professional is a trademark of Capilano Computing Systems Ltd. Osmond PCB is a trademark of Joe Chavez. This document copyright 2007-2008 by Capilano Computing Systems Ltd. All rights reserved.

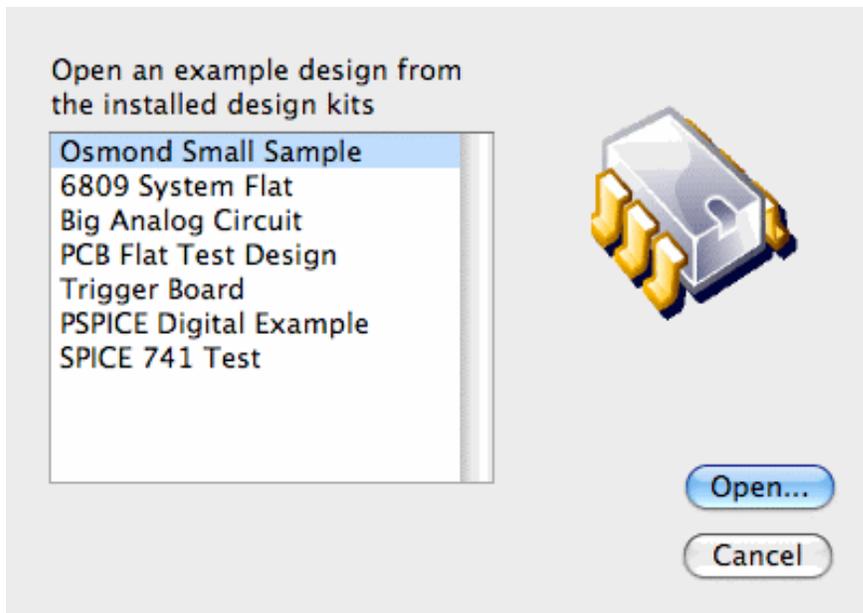
Design Example

Let's start off by running through a simple example.

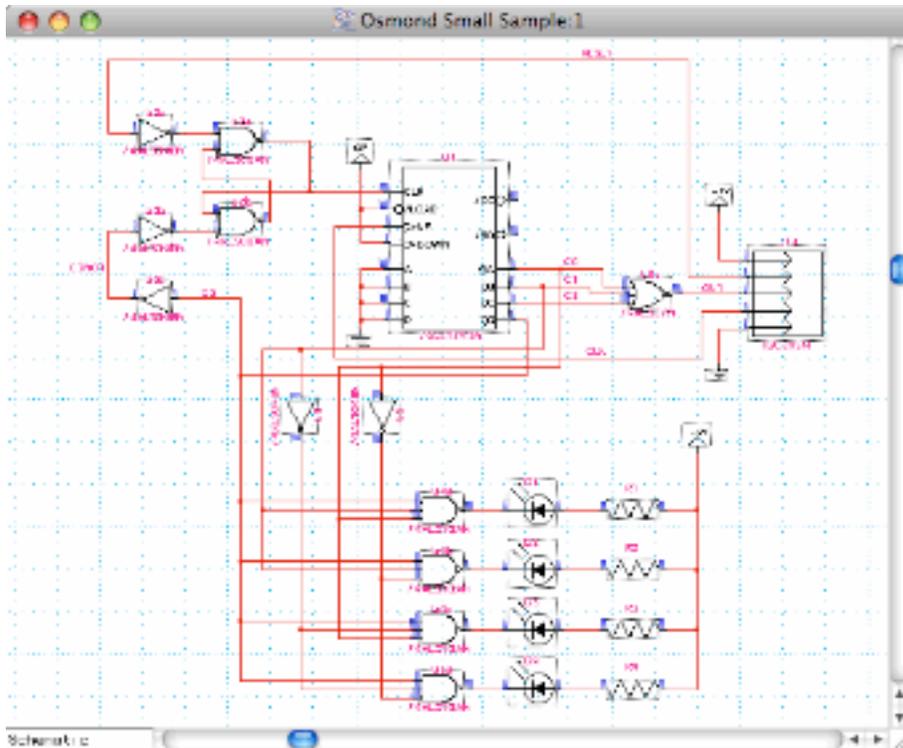
Start up DesignWorks and click on the "Examples" button in the dialog that appears.



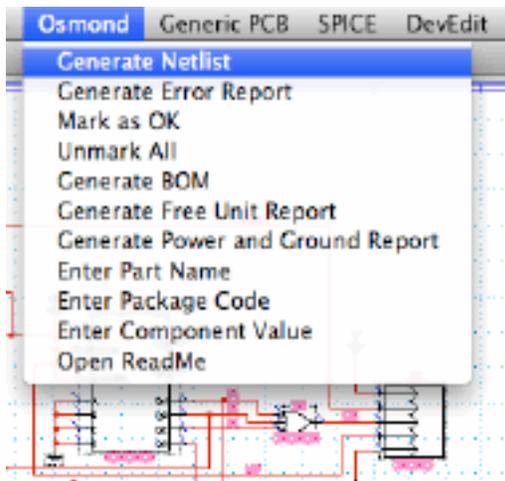
Now select “Osmond Small Sample” and then click on the “Open” button.



You should end up with a window looking like this.



The next step is to generate a file (netlist) that Osmond can read. Go to the “Osmond” menu and select “Generate Netlist”.



This will create a text file in the same folder as the Osmond example file. At this point you could start up Osmond PCB, read in the netlist, and then start the PCB layout process. But before we do that, lets go over what we have done so far.

In order for Osmond PCB to lay out the above design it needs two types of information: A list of what is connected to what and a list of what PCB footprints to use. The connection information is easy, it is created as you build the circuit.

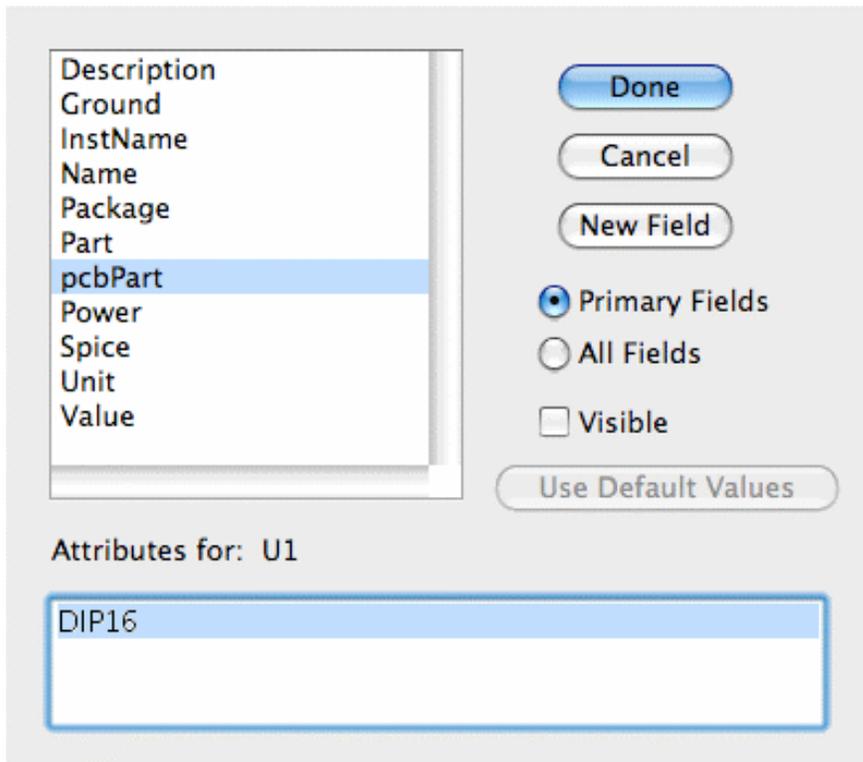
The footprint information is a little more difficult as it requires that you decide what footprint is appropriate for each device. The footprint name is specific to each board layout system. It would be a lot easier if all the PCB layout companies agreed on a common naming scheme, but no such luck.

Note – An easy way to determine what footprints are available in an Osmond library is to open the library with a text editor.

Ok, so you have looked at the Osmond libraries and determined a footprint name, now what? This is where attributes come in. Each device (and other type of circuit element) can have attributes associated with it. In the case of Osmond, the import attribute is the “pcbPart” attribute. This is where you will put the footprint name. Let’s have a look at one.

Go back to the Osmond example file and control click on U1 and select Attributes from the drop down list.

You should see the following dialog:



In this case it is a simple DIP16 footprint, but if we wanted surface mount then the name would change.

As we stated earlier, the interface between DesignWorks and Osmond is a text file called a netlist. The netlister (Scripter) runs through all the devices in the

design and digs out the “pcbPart” attribute and writes it to the netlist file. It also writes out the connectivity information.

The following is a small portion of a netlist file:

```
Part DIP14 { Name U5 }
Part DIP14 { Name U6 }
Part RCR05 { Name R4 }

Signal "Plus5V"
  { J1-5 R1-1 R2-1 R3-1 R4-1 U1-4 U1-11 }
Signal "C0"
  { U1-3 U3-11 U4-11 U4-13 U6-1 }
Signal "C1"
  { U1-2 U3-9 U4-2 U4-4 U6-2 }
```

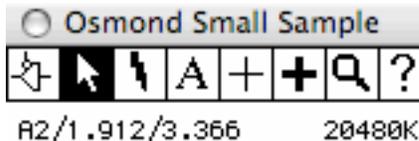
The first part is where the devices and the footprints are associated with each other. The device named U5 has a footprint of “DIP14”. In a perfect world you would never see the contents of the netlist file. In reality you will probably open it on different occasions to verify the information.

Checking the “pcbPart” attribute is going to be something you do quite often and having to bring up the Attribute dialog for each device would be quite painful.

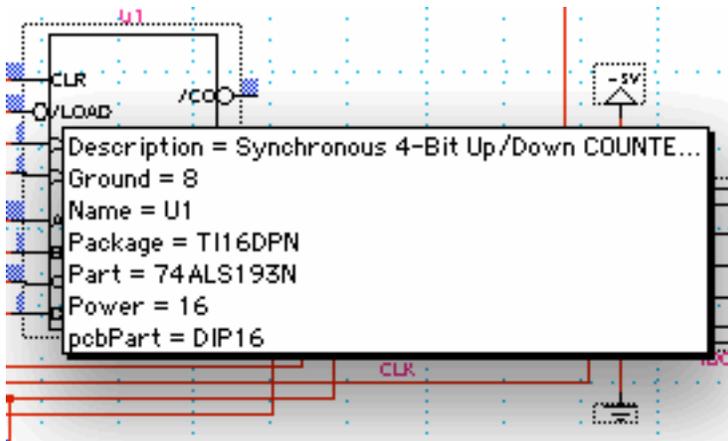
Fortunately there are two other ways to check attribute values.

Method 1 - The Question Mark (?) Tool:

Click on the “?” tool in the floating tool palette



Once the cursor changes to a “?”, click on a device. A small window will popup and stay up as long as you hold the mouse down.



Method 2: Browser Tool

Another way to view attributes is by using the Browser tool. Select “Browser” from the “Tools” menu.

Attribute Browser

Level: Osmond S...

Objects: Devices

Scope: Circuit

Mode:

Browse Edit

Attributes:

Pseudo Primary Sec

Attributes

Description

Ground

InstName

✓ Name

Package

Open Internal

UnSelect All

Show Select

	Name	pcbPart
1	D1	RCR05
2	D2	RCR05
3	D3	RCR05
4	D4	RCR05
5	J1	COND25P
6	R1	RCR05
7	R2	RCR05
8	R3	RCR05
9	R4	RCR05
10	U1	DIP16
11	U2	DIP14
12	U2	DIP14
13	U3	DIP14
14	U3	DIP14
15	U3	DIP14
16	U3	DIP14
17	U3	DIP14
18	U4	DIP14
19	U4	DIP14
20	U4	DIP14
21	U5	DIP14
22	U6	DIP14

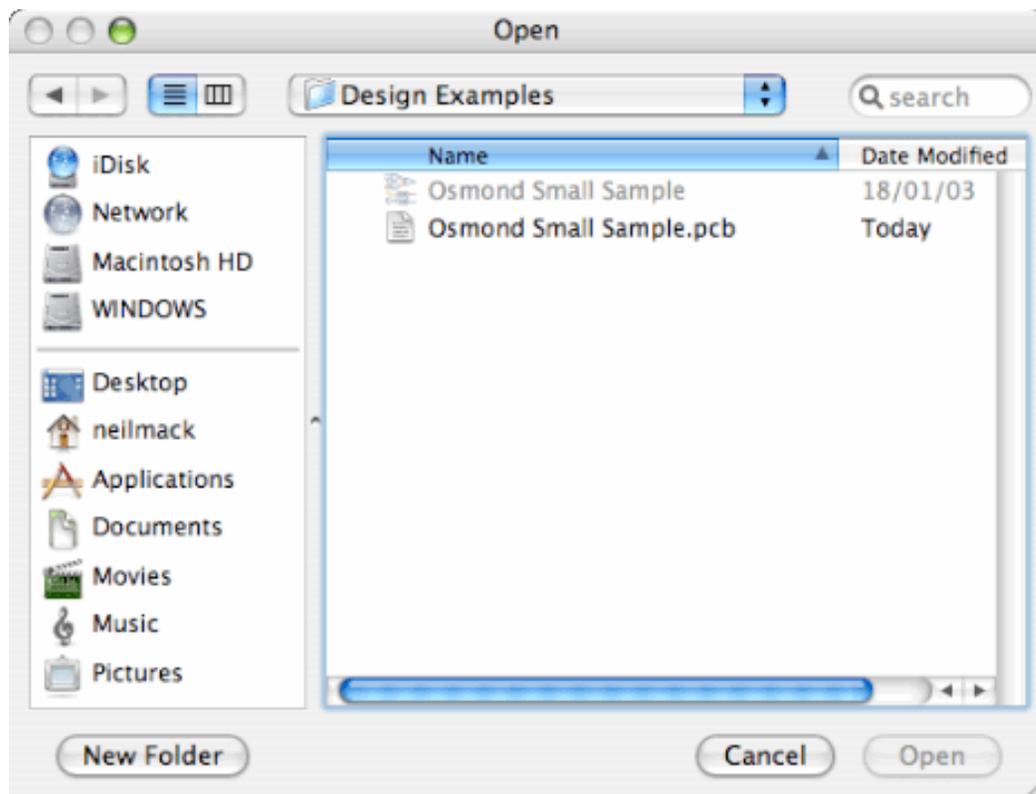
For more information on using the Browser please see the DesignWorks manual.

Now it is time to read in the netlist and start the board layout process.

- Double click on Osmond application and except the standard defaults by clicking on the “OK” button in the window that comes up
- Go to the “File” menu and select “Import” followed by “Library”. A file dialog box will appear, navigate to the Osmond “Tutorial” folder and select “library”.

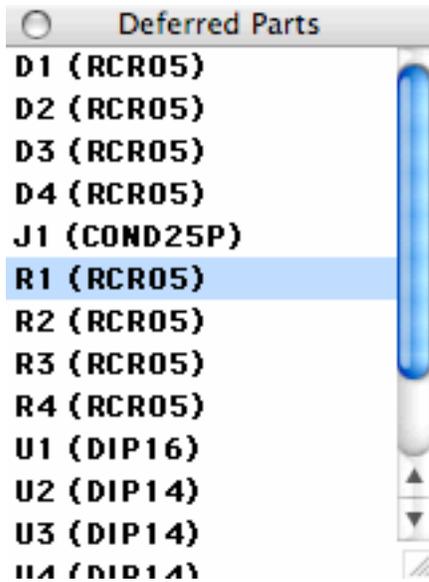
You have just imported a footprint library. Now it is time to import the netlist.

- Go to the “File” menu again but this time select “Part and Net List”. When the file dialog comes up navigate to the netlist file you created earlier in DesigWorks.

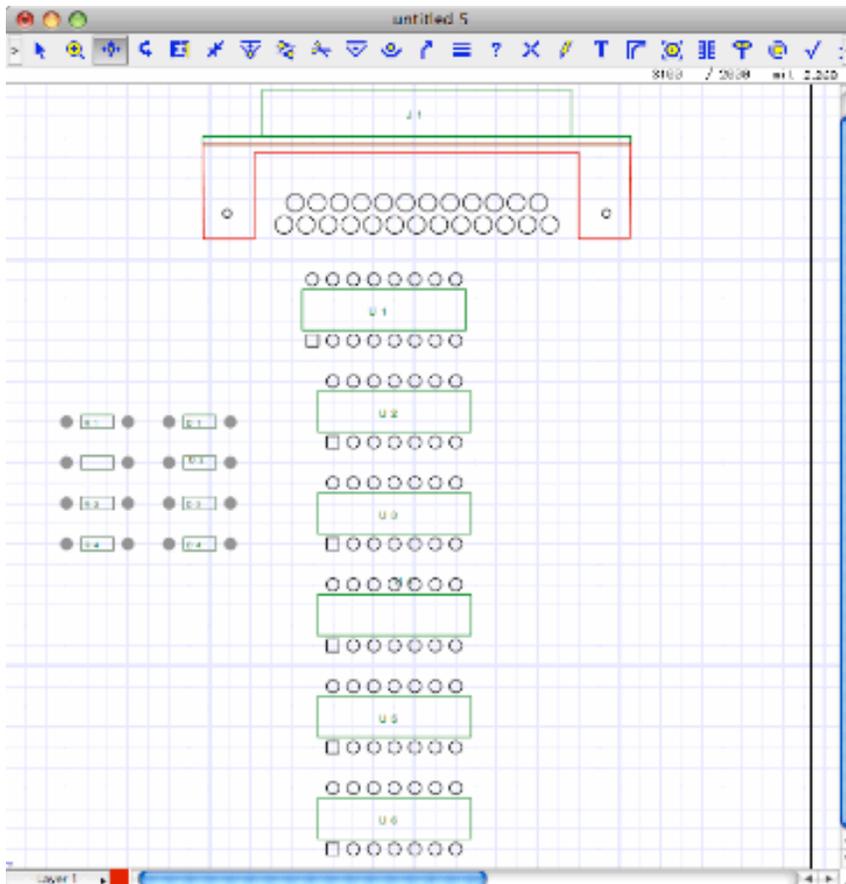


Once you have opened the file you are ready to start placing parts.

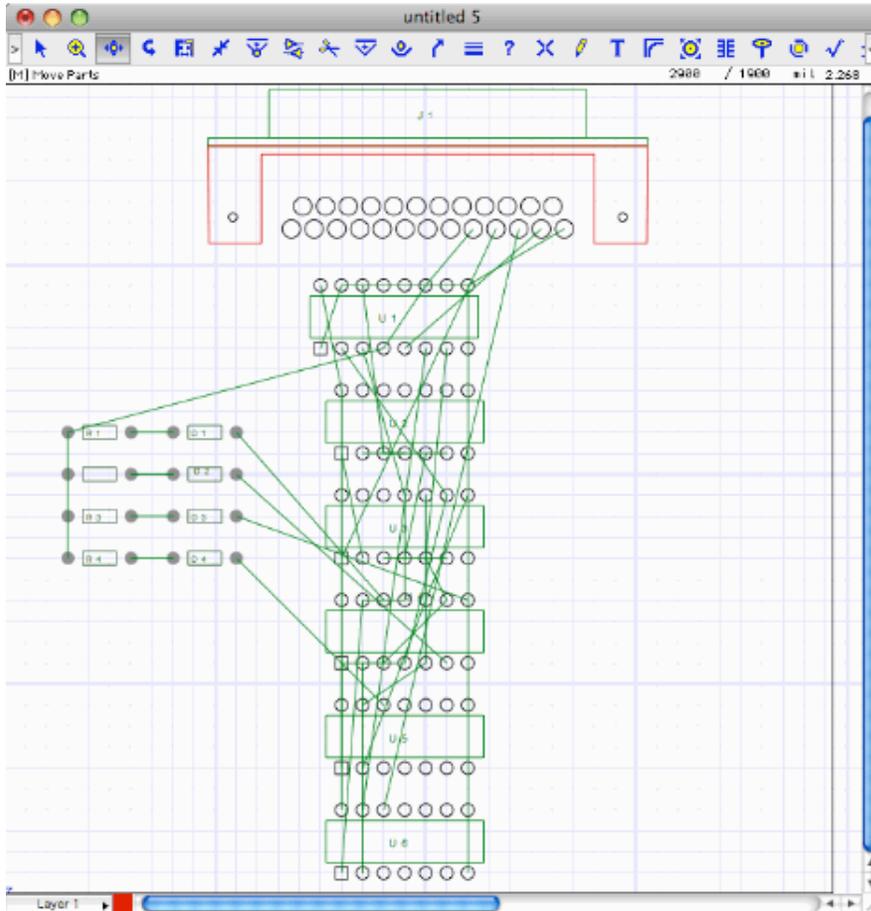
- Go to the “Window” menu and select “Deferred Parts”. A small window will appear.



You can now start dragging parts from this window to the design window. When you have finished you should end up with something like this (or better).



Now go to the “Design” menu and select “Make Rats Nest”. You should end up with something like this:



This short tutorial demonstrates that you can:

- 1) Create a design in DesignWorks
- 2) Export the netlist
- 3) Read it into Osmond
- 4) Lay down footprints
- 5) Generate a rat's nest

The layout of the traces is left as an exercise for the user!

Glossary

Attribute

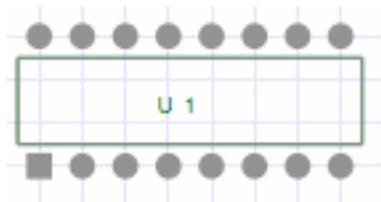
A piece of text associated with a circuit element on the schematic. In DesignWorks, a device can have any number of attributes which contain information like the device name, the name of the library it came from, the package (footprint) code assigned by the user and other information. DesignWorks has numerous attribute fields built in for common needs and also allows you to create your own for your own use that will be stored with the design and can be exported in a netlist or report.

Device Type/Part/Symbol

In DesignWorks they all mean the same. It is the graphical representation of an IC in a schematic editor. There is not always a one to one correspondence between a symbol and an IC. In the case of an “AND” gate the symbol only represents a portion of the IC. Typically you will find 4 “AND” gates in an IC (7400).

Footprint

The physical area an IC takes up on a printed circuit board. Looks something like this:



PCB layout packages come with a library of commonly used package footprints and each footprint has a name. In order to generate a complete netlist, the name of the footprint must be stored with each device on the schematic.

Name/Reference Designator

As each device is placed in a schematic editor (like DesignWorks) it is given a name like “U1”. This is sometimes referred to as a “Reference Designator”, or, for people that don’t like jargon, a “name”. In designs representing a PCB layout, the reference designator is also usually the name of the package on the PCB.

Netlist

A computer file containing information required by a printed circuit board layout system, such as how all the pins are interconnected.

Rat's Nest

This is something you don't want to find under your house, but in a PCB package, it is a display of all the connections shown in the netlist drawn with lines straight from the source to the destination and no attempt to route around other objects. This is done to help visualize where connections are most crowded and how to best lay out the devices.

Signal or Net

In a schematic editor, you draw lines connecting different pins together, forming a common electrical connection. It is the schematic version of a "Trace".

Trace

In a PCB system the copper path that makes an electrical connection between different pins is sometimes called a "Trace". A "signal" on the schematic usually corresponds to a trace on the PCB.