

Netlist File Format

Netlists can be imported into the ExpressPCB layout program using the format described in this document. The netlist file is an ASCII text file. The filename should include the ".net" extension.

Overview:

The netlist is written in a single file, but includes four sections: 1) A file header, 2) A table listing each of the components, 3) A table listing each of the net names, 4) A table listing each of the net connections. Every table entry is written using a single line of text that ends with a CRLF. The fields of the table are separated with *Space* characters (0x20). String fields begin and end with double quotes. Each of the three tables are terminated by a blank line (CRLF).

File Header:

The *File Header* includes 8 entries, each written as a single line of text that ends with CRLF. Strings are enclosed in double quotes. These entries are defined as follows:

- Header Name: String - Should always read "ExpressPCB Netlist"
- Application Name: String - The name of the application that created the netlist file.
- Netlist Format Number: Integer - Should be 1
- Netlist Warning Count: Integer - Should have a value of 0 if the netlist includes no warnings, otherwise the number of warnings. Warnings are recommended to be correct but not required.
- Netlist Error Count: Integer - Should have a value of 0 if the netlist includes no errors, otherwise the number of errors. Errors must be corrected.
- ReservedLine1: String - Should be ""
- ReservedLine2: String - Should be ""
- ReservedLine3: String - Should be ""

An example of the *File Header* section might look like:

```
"ExpressPCB Netlist"  
"SwCAD III version 2.03r"  
1  
0  
0  
"  
"  
"
```

Part IDs Table:

The *Part IDs Table* is a list of Part IDs and their values. For example the Part ID for a resistor might be *R12* with a PartName of *33K*. The table begins with the line of text "*Part IDs Table*", followed by one line for each entry in the table. Each entry includes these three fields:

- Part ID: String - The *Part ID* field identifies a part. It is used to mate components on the schematic with the component footprints placed on the PCB layout. Examples of Part IDs are: *U1*, *R3*, *Battery*. All gates that are in one package must have the exact same Part ID. The ExpressPCB program will consider *U1A* and *U1B* to be in two different packages. The string can include any printable ASCII characters (0x20 through 0x7e) excluding the double quote character (0x22). The string length must be between 1 and 249 characters.
- Part Name: String - This field describes a part. Examples are: *1N4001*, *3K*, *74LS00*, *0.1uf*. The string can include any printable ASCII characters (0x20 through 0x7e) excluding the double quote character (0x22). The string length must be between 1 and 249 characters
- Package Name: String - Current not used and should be: ""

An example of a *Part IDs Table* might look like:

```
"Part IDs Table"  
"R1" "33K" ""  
"U12" "LM741" ""  
"C3" "0.1uf" ""
```

Net Names Table:

The *Net Names Table* is a list of all of the net names. Each net name is listed just once. The table begins with the line of text "*Net Names Table*", followed by one line for each entry in the table. Each entry includes these two fields:

- Net Name: String - The net name identifies a net of connections. The name can be a synthesized one (such as N0001) or a user defined one (such as Gnd, +5v or DataBit0). The string can include any printable ASCII characters (0x20 through 0x7e) excluding the double quote character (0x22). The string length must be between 1 and 249 characters.
- First Net Connection: Integer - Index into the *Net Connections Table* of this net's first connection. A value of 1 indexes to the first entry in the *Net Connections Table*.

An example of a *Net Names Table* might look like:

```
"Net Names Table"  
"N002" 2  
"Gnd" 3  
"+5v" 1
```

Net Connections Table:

The *Net Connections Table* is a linked list of connections. Each entry in the table establishes a connection between a named net, and one pin of a component. The table begins with the line of text "*Net Connections Table*", followed by one line for each entry in the table. Each entry includes these four fields:

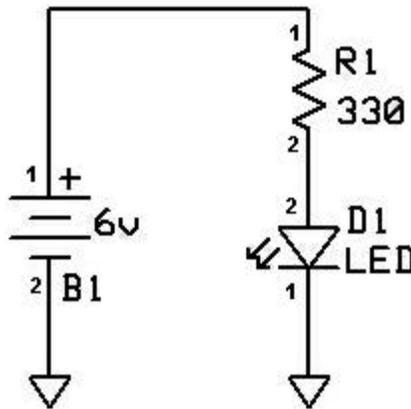
- Net Name Table Index: Integer - Index into the *Net Names Table* for the name of this net. A value of 1 indexes to the first entry in the *Net Names Table*.
- Part ID Table Index: Integer - Index into the *Part ID Table* for the part that this connection is made to. A value of 1 indexes to the first entry in the *Part IDs Table*.
- Pin Number: Integer - The pin number on the part that this connection is made to. Note, the pin number field must be a integer greater than 0, letters are not allowed.
- Next Connection: Integer - Index into the *Net Connections Table* of the next connection that is part of this net. A value of 0 indicates this entry is the last connection. A value of 1 indexes to the first entry in the *Net Connections Table*.

An example of a *Net Connections Table* might look like:

```
"Net Connections Table"  
1 2 40 2  
2 3 2 0  
1 2 39 2  
2 3 1 0
```

Example:

This example shows a schematic along with the netlist that it would generate:



```
"ExpressPCB Netlist"  
"SwCAD III version 2.03r"  
1  
0  
0  
"  
"  
"  
"Part IDs Table"  
"B1 "6v" ""  
"R1 "330" ""  
"D1" "LED" ""  
  
"Net Names Table"  
"N0001" 1  
"Gnd" 3  
"N0002" 5  
  
"Net Connections Table"  
1 1 1 2  
1 2 1 0  
2 1 2 4  
2 3 1 0  
3 3 2 6  
3 2 2 0
```