

CadPack

Import from Orcad

Software tool for import part/net list from Orcad

Technical Info

Version : 2
Code : 81190410.080



SPEA SpA
16, Via Torino
10088 Volpiano - Italy
Tel.: + 39 011 9825 400
Fax: + 39 011 9825 405
E-mail: info@spea.com
Web: www.spea.com

Contents

Introduction	II
1. Orcad file data	1
1.1 Part List.....	2
1.2 Net List.....	3
1.3 Coordinates and access list.....	4
1.4 Wiring and Routing list.....	4
2. Orcad file generalities	5
3. Orcad file format	6
4. Import setting	7
4.1 Pin function assignment	7
4.2 Drawing ref. initials/device type assignment	7
5. Component Properties Identification	8
6. Component properties default value	9

Introduction

CAD files are the base for the automatic generation of test program for InCircuit of any technology.

In order to generate the ICT test program in a short time and without errors, both Bed of Nails and Flying Probe testers require the circuit information available in CAD format.

The Import from Orcad software tool converts the CAD files of the board from the Orcad format to the SPEA board data format.

Conventions, symbols and abbreviations

In the document, the ⓘ symbol is used to highlight information or notes useful to the reader.

Registered trademarks

SPEA is a registered trademark of SPEA SpA.

All other product and company names are trademarks or trade names of their respective companies.

This manual can be updated in accordance with the evolution of the system and associated software. It may contain preliminary contents or it may not be entirely updated with the latest versions used in the system.

Any remarks on errors and imperfections, or suggestions, can be addressed to:

SPEA SpA
Ufficio Documentazione
16, Via Torino
10088 Volpiano – Italy
Tel.: +39 011 9825400
Fax: +39 011 9825405
Email: info@spea.com
Web: www.spea.com

1. Orcad file data

With the “Orcad CAD files” words we refer to the output information generated by the Orcad CAD-CAE programs for the electrical diagrams design and PCB development, used to develop a test application (test program and adapter design).

Information stored in the “Orcad CAD files” concern an electronic board and can be used by an appropriate program to generate a test program and its test adapter design (Bed of Nails or list of movement for Flying Probes).

Information can be grouped in 4 different categories and typically are related to the printed circuit:

Part List
It is the list of all used devices, it must contain: devices drawing reference, part numbers, value, tolerances, device type, etc.
Net List
It is also called wiring list, containing device interconnection data; basically it is presentation of the electrical diagram.
Coordinate and access list
It is the list containing the devices coordinates, concerning their barycentre and pins.
Wiring and Routing list
It is the list containing the path of the Net tracks in the PCB.

For the import of the information above mentioned SPEA has developed the specific program for the translation, stored in a specified format, to its common data bank called “Board Data”.
The name of this type of program is “CAD import driver”.

For the required information, see the list in the following paragraphs.

1.1 Part List

The Part List is an ASCII text file, containing the list of all the parts used to assemble the board; sometimes it can be called **Bill of Material** (BOM).

In the Part List all information concerning the mounted and not mounted parts must be present.
For every part the following information must be defined:

Information	Description
Drawing Reference	Reference designator (e.g. U10, R105, D23, etc.).
Part Number	Device code (e.g. 132549.012, C4QW08, 001-58-AA, etc.).
Value	Device value (e.g. 10K Ω , 10 μ F, 1mH, etc.).
Tolerance	Positive and negative device tolerances (e.g. 1%, 5%, etc.).
Mounting side	The legal values for this item can be: <ul style="list-style-type: none">- Top (Component side)- Bottom (Soldering side)- Not mounted Top- Not mounted Bottom
Rotation ¹	Device mounting rotation angle (e.g. 0°, 180°, etc.).
Dimensions ¹	Device dimensions.
Case code ¹	Device package (case) code.

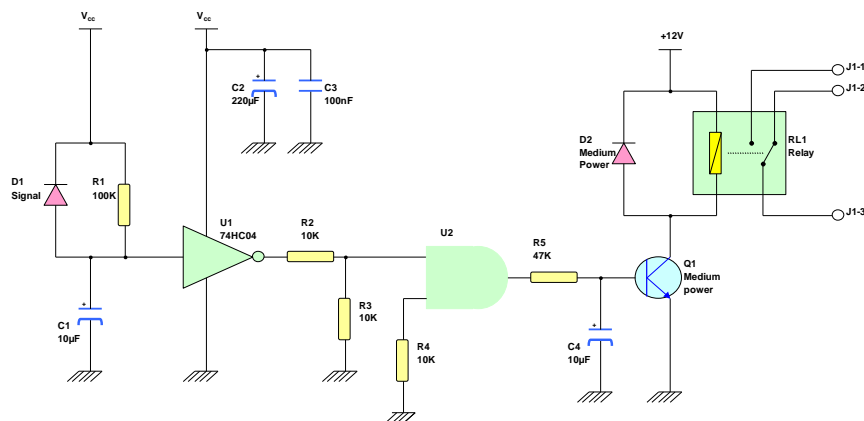
¹ Optional data (not yet managed)

1.2 Net List

The Net List is an ASCII text file containing the device interconnection data; it is also called wiring list. This list must contain the interconnection between devices, including pad and via. Basically, it is the representation of the electrical diagrams.

For every net, the following information must be defined:

Information	Description
Net name	Net identifier (e.g. +5V, RESET, A01, etc.).
Drawing reference	Reference designator of the device connected to the net (e.g. U10, R105, D23, etc.).
Pin name	Name of the device pin connected to the net (e.g. 1, 15, Anode, K, Negative, etc.).
Pin access side	Access side for the device pin, legal values are: <ul style="list-style-type: none"> - Top (Device side access). - Bottom (Soldering side access). - Not accessible - All (both top and bottom side access)



1.3 Coordinates and access list

The Coordinates and access list is an ASCII text file containing the devices coordinates concerning their barycentre and pins. Below, the required information:

Information	Description
Drawing Reference	Reference designator of the device connected to the net (e.g. U10, R105, D23, etc.).
Pin name	Name of the device pin connected to the net (e.g. 1, 15, Anode, K, Negative, etc.).
Pin X position	Pin X-coordinate.
Pin Y position	Pin Y-coordinate.
X barycentre ¹	Device X barycentre.
Y barycentre ¹	Device Y barycentre.

1.4 Wiring and Routing list

The Wiring and Routing list is an ASCII text file that contains all the coordinates of the Net tracks on the PCB and the link with the Net List. So the path of each net on the PCB is described in this file.

For every net the following information must be defined:

Information	Description
Net name	Net identifier (e.g. +5V, RESET, A01, etc.).
X Start	Track segment start X-coordinate.
Y Start	Track segment start Y-coordinate.
X End	Track segment end X-coordinate.
Y End	Track segment end Y-coordinate.
Width	Net segment thickness.
Layer	Layer the segment belongs to.

Example:



¹ Optional data

2. Orcad file generalities

The Orcad required files are: **<FileName>.PAR**
 <FileName>.NEC

Containing the Part list and the Net list.

The SPEA system is based on a PC platform operating in MS-Windows environment.

The files need to be stored into a directory defined by the user.

The SPEA Import from Orcad software tool can retrieve the Orcad file from each defined disk and directory.

3. Orcad file format

The Orcad files listed below are a significant example of the format that the file must have to be converted into SPEA format.

Demo.PAR

Bill Of Materials		January DD,YYYY	hh:mm:ss	Page1
Item	Quantity	Reference	Part	
1	3	C1,C2,C3	1uF 10	Drawing reference
2	3	C4,C6,C12	100nF 10	
3	2	C7,C18	10uF 20	
4	4	R1,R3,R4	10K 5	Value and Tolerance or Device name
5	14	D1,D2,D3,D4,D5,D6,D7, D8,D9,D10,D11,D12,D13, D14	1N4148	
6	3	U1,U2,U3	74LS00	

Demo.NEC (Net list is Computervision format)

0001	VCC	C1-2	C2-2	C3-1
0002	GND	R1-1	R2-1	
0003	A9	R1-1	C2-1	
0004	N0004	U1-1	U2-8	D1-1
0005	SIG1	U1-2	U2-7	D2-1
0006	SIG2	U1-3	U2-6	D3-1
0007	SIG3	U2-5	U1-4	D4-1
0008	AA1	U2-4	U1-5	D3-2
0009	AA3	U2-3	U1-6	D4-1

4. Import setting

4.1 Pin function assignment

This assignment table must be filled, in order to execute correctly the CAD file import.

In order to test correctly some polarized devices such as diodes, bipolar transistors, etc., it is basic to identify correctly the pin function (i.e. anode, base, etc.) of each pin.

The fields contained in the table, are described below:

Field	Description
Device Type	Identifies the type of device (example: Resistors, Capacitors, Digital Devices, Diodes etc.).
Pin Function	Function concerning the Pin.
Pin Name	Pin reference.
Cad Pin	Pin reference in Cad file.

4.2 Drawing ref. initials/device type assignment

The Orcad file typically contains all information about the devices, such as value, tolerances and type; which are fundamental from the test program generation point of view.

The fields contained in the table are described below:

Field	Description
Drawing Reference	Initial letter identifying the Device Type .
Device Type	Identifies the type of device (example: Resistors, Capacitors, Digital Devices, Diodes etc.).
Default Tol+, Tol-	Value and tolerance of the device only if required (as for resistors).

It could happen that in the CAD file they are missing. For each drawing reference initial, the displayed table enables to define the following data default values:

- ◆ Device type
- ◆ Default positive tolerance
- ◆ Default negative tolerance

This means that if, for any reason, the CAD file does not contain the information mentioned above, the default values will be used.

5. Component Properties Identification

The ATPG Software of the SPEA Systems requires to identify the following data for each component:

Passive Components:

- ◆ **Component Family**
- ◆ **Part Number**
- ◆ **Component Value**
- ◆ **Tolerance + and -**

Other Components:

- ◆ **Component Family**
- ◆ **Part Number**
- ◆ **Device Name** (commercial name)

The **Component Family** is not specified in the Part List file so it is necessary to fill a table containing the assignment between Drawing Reference initials and Component Family and the CAD Type before executing the import process.

The table contains also the default tolerance for the specified family of the components.

Example:

Device Type	Drawing. ref. initials	Default Tol+	Default Tol-
Capacitor	C	20	20
Resistor	R	10	10
Connector	J		
Digital IC	IC		

For polarized components such as diodes, it is important to identify the pin function (e.g. Anode) of each pin.

Before running the import it is required to edit the pin Id/pin function table.

Example:

Device Type	Pin Function	Pin Id
Diode	Anode	1
Diode	Cathode	2
Polarized Capacitor	Positive	1
Polarized Capacitor	Negative	2

6. Component properties default value

The SPEA Import software automatically assigns a default value if all or part of the component properties are not available in the CAD file.

In this case a further manual ending can be done to perform the required modifications by using the Board Data editor.

The default values are shown in the following table:

Property	Default Value
Component Family	Not identified
Value of component	0
Tolerance	0
Device Name	None