CadPack

Import from Academi

Software tool for import from Academi Cad format

Technical Info

Version : 2
Code : 81190398.194

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

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 Academi CAD import driver allows to import data present in the Academi CAD file and to convert them in 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. Academi file data

With the “Academi CAD files” words we refer to the output information generated by the Academi 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 “Academi 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 concern 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 mentioned above, 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. 10KΩ, 10µF, 1mH, etc.).
Tolerance	Positive and negative device tolerances (e.g. 1%, 5%, etc.).
Mounting side	The legal values for this item can be: - **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.

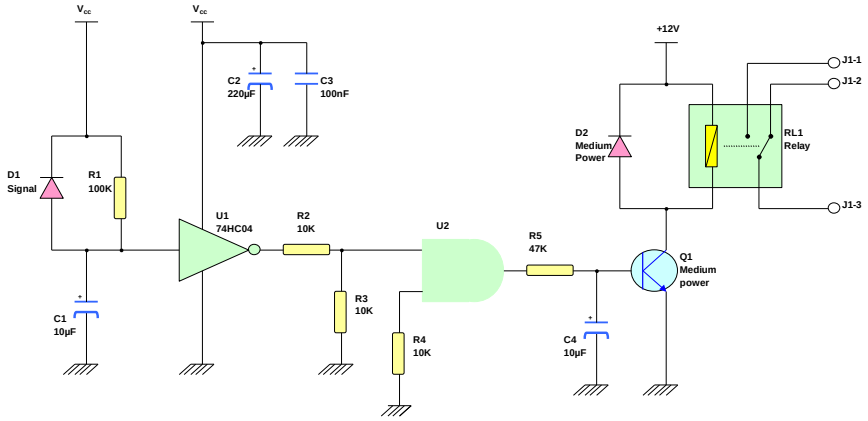
[1] **Optional data (not yet managed)**

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: - **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.

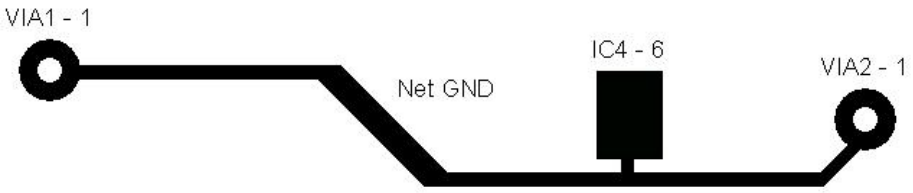
1.4 Wiring and Routing list

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. Academi file generalities

2.1 Academi file name

The files name extensions can be:

A) **.ALL**

B) **.PAR**
.WIR
.LIB
.ART
.JOB

The Academi Neutral file name has to have **".ALL"** extension; it is an ASCII text file and it contains the information concerning the board, component and their connections.

In case of files with the extensions listed at point **B** (see above): in order to create the **".ALL"** file, follow the steps suggested below:

1	Create a new blank text file with **.ALL** extension
2	Copy the contents of each file received into the file with **.ALL** extension
3	Check that all sections are separated by the **.EOD** special string
4	The columns must be separated by one or several blanks (see following example): Example: U9 75176 FSO8 4.675 3.975 3 T U10 MAX232 FSO16L 3.350 3.800 3 T U11 LM 7805 TO220V 1.050 2.450 1 T U12 LM7805 TO220V 2.400 3.425 2 T U13 74HC4040 FSO16 3.175 3.275 3 T U14 MC14490 DILS16 3.750 2.750 3 T The blank between "LM" and "7805" must be deleted or replaced with "underscore" (LM7805 or LM_7805)

2.2 Academi file conversion from Unix to MS-DOS

When the diagram entry has been performed and checked on the Academi CAD workstation, the Academi file **.ALL** should be made available for the SPEA system.

The SPEA system is based on a PC platform operating in a Windows® environment, this means that the CAD import driver can manage ASCII Text file in MS-DOS format.

Due to the fact that the Academi workstation typically uses the Unix operating system, the output ASCII text file has to be converted from Unix to MS-DOS format.

In order to perform the conversion, please refer to appendix A – **Note about the Academi ASCII text file format**.

3. Academi file format

This is a partial extract of an example of a Academi output ASCII text file:

```
DEMO.ALL

:FIL=JOBDATA.DAT
TITLE "COMM.12659 DEMO"
VERSION " "
AUTHOR " "
:EOD

……

:FIL=OUTLINE.LIB>CONN2.OTL>OUTLINE.DAT
16.300  16.300   60   300     1     2   16.369  16.300   0  16.232  16.300   0
:EOD
:FIL=OUTLINE.LIB>DIP8.OTL>OUTLINE.DAT
16.300  16.300   60   300     2     8   16.150  16.450   0  16.150  16.350   0
16.150  16.250   0  16.150  16.150   0  16.450  16.150   0  16.450  16.250   0
16.450  16.350   0  16.450  16.450   0
:EOD
:FIL=OUTLINE.LIB>SMD001.OTL>OUTLINE.DAT
16.300  16.300   40   200     3     2   16.240  16.300   1  16.360  16.300   1
:EOD
:FIL=OUTLINE.LIB>SP0082.OTL>OUTLINE.DAT
16.300  16.300   60   300     1     2   16.000  16.300   0  16.600  16.300   0
:EOD
:FIL=OUTLINE.LIB>SPEC11.OTL>OUTLINE.DAT
16.300  16.300   60   300     1    10   16.100  16.200   0  16.200  16.200   0
 0.000   0.000   0   0.000   0.000   0  16.500  16.200   0  16.500  16.400   0
 0.000   0.000   0   0.000   0.000   0  16.200  16.400   0  16.100  16.400   0
:EOD
:FIL=OUTLINE.LIB>FIDU.OTL>OUTLINE.DAT
:EOD
:FIL=OUTLINE.LIB>FIDU.OTL>LAYER00.AWL
PAD      1 54      16.300 16.300
:EOD
……

:EOD

:FIL=PARTS.PRT
W185                TEST          5.550     5.200   0.0 T
SG1       SG160     SPEC60        5.925     5.525   0.0 T
PZ8       RR        SP0165        6.400     5.550 270.0 T
VA5       -S14K250  SPEC52        7.025     5.625 270.0 T
ME2       FISS.     TORRET        7.047     6.102   0.0 T
C46       680p      SM0805        6.825     6.825 180.0 T
R101      20K       SM0805        6.825     6.950 180.0 T
C37       10uF_25V  SP0100        6.825     7.250 180.0 T
D17       BAS85     M-MELF        7.000     7.250 270.0 T
R87       2K2_1_4W  SP0063        7.125     7.250 270.0 T
ME103               FIDU         12.598     5.354   0.0 T
……

:EOD

:FIL=WIRING.WIR
N00001          04 R40.1     R41.1     IS2.2     W58.1
IPKFDB/1        04 IS8.6     IS5.6     IS2.6     W74.1     R60.1     IS10.6
IPKFDB/1        04 U14.7
%U+5V/2         04 R128.2    R126.1    W173.1    U13.5
%U+15V/2        04 R129.2    R130.1    U13.7     W171.1
%U-15V/2        04 R125.2    W166.1    R122.2    U13.8     R123.1
OUTERR/3        04 IS12.1    R67.1     D41.1     R66.2     U8.1      W73.1
VCC3842/5       04 R80.2     C35.1     U11.11    W108.1    D46.2     R106.1
VFLY/5          04 TH1.8     W147.1    R82.1     C52.1     D55.2     PD2.1
GNDPOWER        11 IS9.5     C22.2     C53.1     C7.1      U7.2      U7.3
GNDPOWER        11 C14.2     C68.2     R7.2      Q7.3      IS6.5     C20.2
GNDPOWER        11 U5.3      U5.2      C13.2     C5.1      C66.2     R5.2
GNDPOWER        11 Q5.3      Q3.3      R3.2      C64.2     C3.1      U3.2
GNDPOWER        11 U3.3      C12.2     C18.2     IS3.5     C10.2     U1.2
GNDPOWER        11 U1.3      C1.2      RL1.2     RL1.5     C17.2     RL1.6
GNDPOWER        11 R50.2     IS10.3    CF1.2     U8.12     R65.2     R68.2
GNDPOWER        11 D45.2     R75.1     R78.2     IS14.3    IS13.3    R90.2
GNDPOWER        11 R88.2     R86.2     R83.1     C37.2     R101.2    C46.2
GNDPOWER        11 U12.4     DZ1.1     PZ5.1     C38.2     CF2.2     DZ2.1
```

```
GNDPOWER        11 C49.2     Q11.1     C54.2     VA2.1     VA3.1     VA4.1
GNDPOWER        11 Q10.1     C51.2     C45.2     C52.2     W159.1    C62.2
GNDPOWER        11 R1.1      C43.2     R97.2     R94.2     U11.8     U11.9
GNDPOWER        11 C35.2     C15.2     PD1.3     TP1.1     Q1.3      C34.2
GNDPOWER        11 C50.2
……
:EOD

……

:EOF
```

The **.ALL** file consists of 3 sections:

- **OUTLINE**
- **PARTS**
- **WIRING**

The unit of measurement is supposed to be in INCH because it is not defined in the file.

In the next paragraphs, a short description for each section is provided.

3.1 PARTS

Basically this section contains the part list and mounting data of each single device present in the Academi CAD file; data are separated by blanks.

Every single row of the Academi file, in this section, contains the following information:

1. **Drawing reference**
2. **Value of Device name**
3. **Package name**
4. **X-Coordinate**
5. **Y-Coordinate**
6. **Rotation**
7. **Mounting Side**

ⓘ **Note: Value or Device name**, if there are, and **PackageName** form the Part Number.

The following example shows the syntax used for the **PARTS** section:

1	2	3	4	5	6	7
Drawing reference	Value or Device name	Package name	X coord.	Y coord.	Rotation	Mounting side
C46	680p	SMO805	6.825	6.825	180.0	T
D17	BAS85	M-MELF	7.000	7.250	270.0	T
W185		TEST	5.550	5.200	0.0	T

Typically, **PARTS** section, as shown in the following example:

```
.....
:FIL=PARTS.PRT
W185                TEST          5.550     5.200   0.0 T
SG1       SG160     SPEC60        5.925     5.525   0.0 T
PZ8       RR        SP0165        6.400     5.550 270.0 T
VA5       -S14K250  SPEC52        7.025     5.625 270.0 T
ME2       FISS.     TORRET        7.047     6.102   0.0 T
C46       680p      SM0805        6.825     6.825 180.0 T
R101      20K       SM0805        6.825     6.950 180.0 T
C37       10uF_25V  SP0100        6.825     7.250 180.0 T
D17       BAS85     M-MELF        7.000     7.250 270.0 T
R87       2K2_1_4W  SP0063        7.125     7.250 270.0 T
ME103               FIDU         12.598     5.354   0.0 T
.....
:EOD
```

3.2 OUTLINE

This section is used to describe the package properties (name, number of pins, type); data are separated by blanks in a row of the Academi file.

The pin coordinates can be extracted using two different identifiers (the second mode is used when the **OUTLINE.DAT** identifier is missing):

- In the **OUTLINE.DAT** subsection:
  1. **Initial Pin offset X**
  2. **Initial Pin offset Y**
  3. Not used
  4. Not used
  5. Not used
  6. **Pin count**
  7. **Pin offset**

  The Package Name is described in a header of each **OUTLINE** section. The pin coordinates are calculated in this way:

  PinX = Initial Pin OffsetX – PinXOffset
  PinY = Initial Pin OffsetY – PinYOffset

  It is referred to the barycentre of the package.

  If the Pin Type is equal to 0, then the Package type is SMD, else if the Pin Type is equal to 1 then the Package type is TH.

  If the PinX and PinY offsets are equal to 0, then this pin is not managed.

  The following example shows the syntax used for the **OUTLINE.DAT** subsection:

1	2	3 - 5			6	7
Initial Pin offset X	Initial Pin offset Y	Not used			Pin count	Pin Offset
16.300	16.300	60	300	1	2	

  The following example shows the syntax used for the **Pin Offset** field:

1	2	3
Pin X offset	Pin X offset	Pin type
16.300	16.300	0
16.232	16.300	0

- In the **LAYER00.AWL** subsection:
  1. Not used
  2. **Pad count**
  3. Not used
  4. **Pad offset**

  The Package Name is described in a header of each **OUTLINE** section. The pad coordinates are calculated in this way:

  PadX = 16.3 – PadXOffset
  PadY = 16.3 – PadYOffset

It is referred to the barycentre of the package.

The Package type is supposed SMD.

The following example shows the used syntax for the **LAYER00.AWL** subsection:

Not used	Pad count	Not used	Pad offset	
			Pad X offset	Pad Y offset
PAD	1	54	16.300	16.300

Tipically, the **OUTLINE** section, as shown in the following example:

```
.....
:FIL=OUTLINE.LIB>CONN2.OTL>OUTLINE.DAT
16.300  16.300   60   300     1     2   16.369  16.300   0  16.232  16.300   0
:EOD
:FIL=OUTLINE.LIB>DIP8.OTL>OUTLINE.DAT
16.300  16.300   60   300     2     8   16.150  16.450   0  16.150  16.350   0
16.150  16.250   0  16.150  16.150   0  16.450  16.150   0  16.450  16.250   0
16.450  16.350   0  16.450  16.450   0
:EOD
:FIL=OUTLINE.LIB>SMD001.OTL>OUTLINE.DAT
16.300  16.300   40   200     3     2   16.240  16.300   1  16.360  16.300   1
:EOD
:FIL=OUTLINE.LIB>SP0082.OTL>OUTLINE.DAT
16.300  16.300   60   300     1     2   16.000  16.300   0  16.600  16.300   0
:EOD
:FIL=OUTLINE.LIB>SPEC11.OTL>OUTLINE.DAT
16.300  16.300   60   300     1    10   16.100  16.200   0  16.200  16.200   0
0.000   0.000   0   0.000   0.000   0  16.500  16.200   0  16.500  16.400   0
0.000   0.000   0   0.000   0.000   0  16.200  16.400   0  16.100  16.400   0
:EOD
:FIL=OUTLINE.LIB>FIDU.OTL>OUTLINE.DAT
:EOD
:FIL=OUTLINE.LIB>FIDU.OTL>LAYER00.AWL
PAD      1 54      16.300 16.300
:EOD
.....
```

3.3 WIRING

This section is used to describe the device pin properties (pin name, net name, drawing reference); data are separated by blank in a row of the Academi file.

The “Import from Academi” import CAD driver manages the following data:

1. **Net name**
2. Not used
3. **Pin connection** (Drawing reference.Pin name)

The following example shows the syntax used for the **WIRING** section:

1	2	3	
Net name	Not used	Pin connection	
		Drawing ref.	Pin name
%U+5V/2	04	R128	2
%U+5V/2	04	R126	1
%U+5V/2	04	W173	1
%U+5V/2	04	U13	5

Tipically, the **WIRING** section, as shown in the following example:

```
.....
:FIL=WIRING.WIR
N00001          04 R40.1     R41.1     IS2.2     W58.1
IPKFDB/1        04 IS8.6     IS5.6     IS2.6     W74.1     R60.1     IS10.6
IPKFDB/1        04 U14.7
%U+5V/2         04 R128.2    R126.1    W173.1    U13.5
%U+15V/2        04 R129.2    R130.1    U13.7     W171.1
%U-15V/2        04 R125.2    W166.1    R122.2    U13.8     R123.1
OUTERR/3        04 IS12.1    R67.1     D41.1     R66.2     U8.1      W73.1
VCC3842/5       04 R80.2     C35.1     U11.11    W108.1    D46.2     R106.1
VFLY/5          04 TH1.8     W147.1    R82.1     C52.1     D55.2     PD2.1
.....
:EOD
.....
```

4. Import setting

4.1 Pin function assignment

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

In order to correctly test some polarized devices such as diodes, bipolar transistors, etc., it is basic to correctly identify 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 Academi file typically contains all information about the devices, such as value, tolerances and type; which are fundamental from the point of view of the test program generation.

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.

A. Note about the Academi ASCII text file format

The Academi CAD-CAE typically runs under Unix operating system and generates its neutral ASCII output file in Unix format.

The Unix ASCII text files use as end of line identifier, the"$0a_{hex}$" ASCII character.
The Windows® (MS-DOS) operating system uses the ASCII characters "$0d_{hex}$" and "$0a_{hex}$"as end of line identifier for ASCII text files.
This means that output ASCII text files may require an ASCII format conversion (from Unix to Windows® format).

This operation can be performed using "WordPad", a standard text file editor.
Open the Academi ASCII file with this editor and save it, this operation will automatically perform the conversion from ASCII Unix format to ASCII Windows® format.