AEGIS

S O F T W A R E

AEGIS Cad Import Guide

Manufacturing Operations Software

Cad Types:

To submit any change request or update to this document please email esupport@aiscorp.com

Aegis CAD Import Guide

1   Altium Designer

Extraction Procedure:
Schematic and PCB design files can be exported into various formats - including previous versions of Altium's software or alternate software packages. Export of files is performed using the **File»Save As** command, or the **File»Save Copy As** command.

Common extension: .PcbDoc
Save as Type: Altium PCB ASCII (*.PcbDoc)
Import: Protel Importer

Note:
Format of CAD file must be Version 3.0 or greater and in ASCII format.
Version: 9.2 supported in Aegis Version 7.6.2.0
Version: 9.3 support due in the Aegis Version 7.6.3.0

Beginning File Header:

|RECORD=Board|SELECTION=FALSE|LAYER=UNKNOWN|LOCKED=FALSE|POLYGONOUTLINE=FALSE|USERROUTED
=TRUE|UNIONINDEX=0|FILENAME=C:\Altium\PROJECTS\2ASCII.PcbDoc|KIND=Protel_Advanced_PCB|VERSION=5.00|D
ATE=7/14/2009|TIME=2:17:54 PM|…….

2   Cadence Allegro

Extraction Procedure:
Cadence Allegro requires the use of a script available from www.aiscorp.com
The script produces a ccam.cad file that can then be imported into CircuitCAM.

Common File Extension: .cad

File header:

A!REFDES!CLASS!SUBCLASS!COMP_DEVICE_TYPE!COMP_PACKAGE!SYM_ROTATE!PIN_NUMBER!DRILL_HOLE_NA
ME!NET_NAME!PIN_X!PIN_Y!START_LAYER_NAME!END_LAYER_NAME!GRAPHIC_DATA_NAME!GRAPHIC_DATA_1!G
RAPHIC_DATA_2!GRAPHIC_DATA_3!GRAPHIC_DATA_4!GRAPHIC_DATA_5!GRAPHIC_DATA_6!GRAPHIC_DATA_7!GR
APHIC_DATA_8!GRAPHIC_DATA_9!GRAPHIC_DATA_10!SYM_NAME!SYM_X!SYM_Y!SYM_MIRROR!
J!D:\cam\102.brd!Tue Apr 25 15:21:27 2000!-100.000!-170.000!500.000!430.000!0.001!millimeters!B01!47.2 mil!6!OUT OF
DATE!
S!!BOARD GEOMETRY!OUTLINE!!!0.000!!!!!!!!!ARC!-8.095!-9.949!-8.095!-9.949!-10.000!-
9.949!1.905!0.000!COUNTERCLOCKWISE!NOTCONNECT!TARGET!-10.000!-10.000!NO!

A Cadence Allegro file can be very easily distinguished by the ! marks in the output file.
29 data fields are required in the output file.

Note:
For Cadence OrCAD it is now recommended to use the IPC-2581 standard #3

3 Cadence OrCAD IPC-2581

Common File Extension: .xml

File header:

```
<?xml version = "1.0" encoding = "UTF-8"?>
<IPC-2581 revision="A" xmlns="http://webstds.ipc.org/2581" xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
xmlns:xsd="http://www.w3.org/2001/XMLSchema">
```

Note:
This is the recommended export if available, otherwise the script method can be used #2

4 Eagle Layout Editor

Extraction Procedure:
Visit www.cadsoft.de – Download - ULPs - User Language Programs
Download the gencad.ulp or fabmst12.ulp which uses the FABMASTER import.

5 EE Designer III ASCII File

Common File Extension: .ala

6 GenCAD v 1.4 from Veribest

Extraction Procedure:
Veribest provides a stand-alone application called Report Writer. Use this application to export the "Mitron" export option, which causes Veribest to produce a GenCAD compliant output ASCII file.

Common File Extension: .cad

File Header:

```
$HEADER
GENCAD 1.4
USER RSI-TRANSLATOR GENCAD OUTPUT V:10
DRAWING scm
REVISION Wed Jan 07 15:13:12 1998
UNITS USER 1000
ORIGIN 0 0
INTERTRACK 0
$ENDHEADER
```

7   Incases TL CAD File

Common File Extension: .tl

8   IPC-D-356

Common File Extensions: .net or .356

File Header:

```
P  JOB  010670x2.brd
P  UNITS CUST
P  DIM   N
C
C  Feb 11 16:26:42 2000
Company.
C  IPC-D-356 Netlist From Allegro
C
327N/C            -        A02X-000460Y+036040X1200Y1200R000 S1
```


9   Mentor Graphics Expedition

Extraction Procedure:
Select Export under the File menu.
Select the General Interface menu item.
Select the Mitron GenCAD option.

Common File Extension: *.cad

File Header:

See GenCAD section

10   Mentor Graphics Neutral File

Extraction Procedure:
Select the "Write Neutral Files" command from the Output menu of the FabLink utility.
Select to output all options.

Common File Extension: *.neu

File Header:

```
# file : /users/ngd/pci_audio/t20318pt1/pcb/mfg/neutral_file.mech
# date : Wednesday June 10, 1998; 13:58:18
#
```

11 OrCAD

Extraction Procedure:
Open the OrCAD Layout Application. DO NOT OPEN the Project or the Board.
Select Export from the File menu.
Select the GenCAD option.
Browse to the location of the file and Accept.

Common File Extension: *.cad

File Header:

See GenCAD section

12 PADS

- PowerPCB
- PADS Perfrom
- PADS 2000
- PADS Work

Extraction Procedure:
Through the In/Out menu, select ASCII Out (F4) command.
Select the "All" option and the PADS-2000 or PADS-3000 format.
Input the name to be used for the output file.

Common File Extension: *.asc

File Header:

!PADS-POWERPCB-V4.0-BASIC! DESIGN DATABASE ASCII FILE 1.0
PCB   GENERAL PARAMETERS OF THE PCB DESIGN

13 PANTHEON PDB File

Common File Extension: none

14 P-CAD

- Accel EDA
- Accel Tango
- Accel PCAD
- P-CAD for DOS
- P-CAD PDIF Design File

Extraction Procedure:

From the File Menu select SAVE AS
Select the ASCII .PCB option

Common File Extension: *.pcb

Extraction Procedure:

From the File Menu select Export
Select the PDIF

Common File Extension: *.pdf

OLD EXTRACTION

1. Select the PCAD menu from main.
2. Select PCB Tools.
3. Select the PDIF File Writer option.
4. Output a complete PDIF data file for the design. (Note that output options are not relevant to CircuitCAM required data. The CAD user may choose any options.)
5. Select the name of the PCB file desired for export.
6. Input the name of the PDIF file to be created for input to CircuitCAM.
7. Click on Run.

Common File Extension: *.pdf

File Header for PDF:

```
%*****************************************************************
%
%    Program  :   ACCEL P-CAD PCB Version 14.01.03
%    Date    :   Jan 28 1999
%    Time    :   08:00:05 PM
%    File In :   c:\accel\demo\modfax0.pcb
%    File Out :   c:\accel\demo\modfax0.pdf
%    Format  :   P-CAD DATABASE INTERCHANGE FORMAT
%
%*****************************************************************
```

File Header for PCB:

ACCEL_ASCII "C:\WINDOWS\Desktop\Htrdemo.pcb"

```
(asciiHeader
  (asciiVersion 3 0)
  (timeStamp 2002 1 17 9 18 44)
  (program "P-CAD 2001 PCB" "16.02.04")
```

```
 (copyright "Copyright © 1991-2001 Altium Limited")
 (fileAuthor "")
 (headerString "")
 (fileUnits Mil)
 (guidString "{AA3EBDF5-111B-48FC-9292-C084E2766C15}")
)
```

15   PROTEL 98/99 ASCII PCB File ver 3

Extraction Procedure:

Within Protel, select the File menu->Save As.
When in the File Format dialog, select ASCII .PCB

Common File Extension: *.pcb

File Header:

|RECORD=Board|FILENAME=D:\Shoebox\PCB_design\SwitchTest\Backup of Copy of
SW_2.PCB|KIND=Protel_Advanced_PCB|VERSION=3.00|DATE=24-May-
2001|TIME=11:52:14|ORIGINX=2050mil|ORIGINY=2250mil|BIGVISIBLEGRIDSIZE=10000000.000|VISIBLEGRIDSIZE=1000
00.000|ELECTRICALGRIDRANGE=0.5mil|ELECTRICALGRIDENABLED=TRUE|SNAPGRIDSIZE=5000.000000|SNAPGRIDS
IZEX=5000.000000|SNAPGRIDSIZEY=5000.000000|TRACKGRIDSIZE=50000.000000|VIAGRIDSIZE=200000.000000|COM
PONENTGRIDSIZE=5000.000000|COMPONENTGRIDSIZEX=5000.000000|COMPONENTGRIDSIZEY=5000.000000|CURR
ENTWORKINGLAYER=BOTTOM|DOTGRID=TRUE|DISPLAYUNIT=1|PLANE1NETNAME=(No Net)|PLANE2NETNAME=(No
Net)|PLANE3NETNAME=(No Net)|PLANE4NETNAME=(No Net)|PLANE5NETNAME=(No Net)|PLANE6NETNAME=(No
Net)|PLANE7NETNAME=(No Net)|PLANE8NETNAME=(No Net)|PLANE9NETNAME=(No Net)|PLANE10NETNAME=(No
Net)|PLANE11NETNAME=(No Net)|PLANE12NETNAME=(No Net)|PLANE13NETNAME=(No Net)|PLANE14NETNAME=(No
Net)|PLANE15NETNAME=(No Net)|PLANE16NETNAME=(No Net)

16   SCI Cards Neutral File

Extraction Procedure:

From CircuitCAM you must extract a SCI-Cards Neutral file.

Common File Extension: *.cii

17   Supermax CAD

Common File Extension: *.ipl

18   Tango for DOS

See PCAD .pcb export.

19 Ultiboard

Current CAD support for Ultiboard is not available. Other exports that might be available are IPC-D356 and Gerber.

Old Extraction Procedure:

Use the the DDF2GenCAD convertor v1.30 supplied by Ultiboard.
Import as a Gencad File

File Header:

See GenCAD

20 Valor ODB++ Cad Project

Extraction Procedure:

A .tar or a .tgz file is normally supplied containing the full directory structure of the Board.
Unzip keeping the directory structure within the .tar file.
To import browse to the MATRIX folder and select the MATRIX file.

Note: This import is primarily for the VALOR application itself.

Common File Extensions supplied by customer: *.tar or .tgz

21 ZUKEN VISULA and CADSTAR

 Extraction Procedure:

In the Job menu, select Interface option.
Select CADIF output from the list.

Common File Extension: *.paf

CADstar is capable of outputting the CADIF file format.

Please use the output option for CADIF and import via the CircuitCAM Visula/CADIF interpreter.

File Header:

```
cadif
 (format CADIF 4 0)
 (design
  (dataSet ARCHIVE XFER_FILE ROUTE_RULES PLACE_RULES)
  (paper (name "B")
   (box (pt -21590000 -13970000) (pt 21590000 13970000))
   (designOrigin (pt 0 0))
```