

# Aegis Software Required Data Forms

A Guide for our customers of providing BOM and CAD Data

The Aegis software system converts your CAD design data and BOM data into a wide array of useful information such as robotic assembly machine programs and quality documentation more quickly and accurately than ever possible with manual methods. More information can be found on the Aegis Industrial Software Web Site: <http://www.aiscorp.com/> or feel free to contact Variosystems.

This document defines the data it requires, and also how to obtain these forms of data. It is divided into four sections:

**Section 1:**

BOM sources for using CircuitCAM only or using CheckPoint's extended support.

**Section 2:**

Data that includes all native CAD importers as well as Gerber and scanned bare board importers.

**Section 3:**

The procedures for extracting the CAD files from the board design software for most CAD sources.

## Section 1: BOM Import Sources and Formats

Aegis developed CheckPoint to address the challenges found in most bills of material. CheckPoint permits users to rapidly import BOMs in various source formats, to verify the BOM's integrity, and to perform both BOM-to-BOM and BOM-to-CAD comparisons. The validated CheckPoint BOM is directly transmitted to the CircuitCAM project file.

### BOM Data Requirements:

In order to successfully process your assemblies with our new CircuitCAM software, we will require a bill of material. However, more data is always beneficial in order to cope with unforeseen file difficulties, so if more information and formats are available, please provide them.

Format	Supported in...		Delimiters	Required Fields	Preferred Fields	Optional Text Fields	Optional Number Fields	Optional Boolean Fields	Custom BOM Fields	Additional Functions in CheckPoint
	CheckPoint	CircuitCAM								
Text (ASCII)	Yes	Yes	Fixed Column or Character	Part Number, Reference Designators	Description, Quantity (CheckPoint)	Part Marking, Package, Population Type, Customer P/N, Mfr, Mfr P/N, Vendor, Vendor P/N, Bin Location, Stock Code, Software Ver, Part Rev, URL	Cost, Height (mils)	Polarity, Socketed (Y/N, Yes/No, T/F, True/False)	Unlimited	Template Definition, Join Multiple Files to Form 1 BOM, Analysis Reports, Use Customer or Internal Number as Primary, AML/AVL Mgmt, Revision Control
Excel	Yes	No	N/A							
ODBC (Access)	Yes	No	N/A							

## Section 2: CAD Import Sources and Formats

### CircuitCAM Data Requirements:

In order to successfully process your assemblies with our new CircuitCAM software, we will require a bill of materials (refer to Section 1) and one of the following groups of board design data. However, more data is always beneficial in order to cope with unforeseen file difficulties.

### Summary Information of Data Sources

Native CAD Data
Accel EDA , Accel Tango, & Accel PCAD
Cadence Allegro via Aegis Script
DIF (Design Interchange Format), PADS DFT Audit, and C-Link
EE Designer III ASCII File
GenCAD v. 1.4
INCASES TL CAD File
Intergraph Veribest GenCAD
IPC-D-356
Mentor Graphics Neutral File
OrCAD
PADS PowerPCB, PADS Perform, PADS 2000, PADS Work
Pantheon PDB File
P-CAD PDIF Design File
P-CAD for DOS
Protel 98/99 ASCII PCB File
SCI Cards Neutral File
Supermax CAD
Tango for DOS
Ultiboard
Valor ODB++ CAD Project
Zuken Visula and Cadstar

3rd Party CIM Software Applications
Fabmaster FATF File
Panapro / PanaCIM PCB File
Unicam Project File

PCB Design Data	Format	Provides	Required?
Gerber Data	RS-274X, RS-274-D	Locations, reference designators, and board graphics	YES
Aperture List	Text File	Shape information required to render the Gerber image	RS-274-D: YES RS-274-X: NO
Centroid File	Text File	Relative locations of components (defined by reference designator) on the PCB. Allows automatic joining of references to the Gerber data.	NO: However, the use of this file expedites Gerber processing.

Assembly Machine Program Data - Suitable For Machine Programming
CONTACT Systems 3AV/S/Z Series SMT Machines PLA Program and Libraries
Europlacer Program Files
Fuji Source Programs and Part Data Files
MYDATA automation PCB Files and TPSYS Databases
Panasonic POS File
PHILIPS FTIP-II Programs
Sanyo Chipshooter Program
SIEMENS Program Files, and Part Databases

## Board Design Data:

All CAD systems output Gerber data. Therefore, you can be assured that at least this data has been available at some time in the product's life cycle. The photoplotters used to fabricate bare boards are driven by this data exclusively, and therefore if a bare board exists, someone had access to the Gerber data at one time.

**Please obtain *at least one* of the sets of data listed below.  
(The sets are listed in order of preference, 1 being most preferred):**

**Set 1: Any of the Listed Native CAD File or 3<sup>rd</sup> Party CIM Software Application Import Formats  
(See Chart above. Section 3 below provides instructions for providing data.)**

### Set 2: Gerber RS-274D or RS-274X Data Files

Gerber Files to include:

- Aperture List(s) (D-code, Shape, X-size, Y-size) \*\*\*\*RS274D ONLY\*\*\*\*
- Top Mask Layer
- Top Silkscreen Layer
- Bottom Mask Layer (if applicable)
- Bottom Silkscreen Layer (if applicable)

**Optional but Highly Desirable:** ASCII text component location file (Ref, X, Y, theta)

### Set 3: ASCII Centroid File

Not essential but helpful if available. Most CAD systems export this file and it is sometimes identified as an "insertion", "manufacturing", or "CAM" file. It can be used as a sole CAD information source, but these files lack graphical information required for proper visual aid and documentation development. The text (ASCII) file contains the following:

- Reference designators
- X coordinate
- Y coordinate
- Package rotation (theta)

**Set 4: Scanned Board Image** – Please provide one of the following:

- Scanned Image – (.jpg or .bmp formats)
- Bare Board
- Assembly Drawing

**Optional but Highly Desirable:** ASCII text component location file (Ref, X, Y, theta)

**Set 5: Any of the Listed Machine File Import Formats (See Chart above)**

## Section 3: EXTRACTING NATIVE CAD DATA FROM PCB BOARD DESIGN SOFTWARE

---

### 1 Cadence Allegro

#### Extraction Procedure:

Cadence Allegro requires the use of a script available from

[www.variosystems.ch/images/Downloads/Cadence\\_Allegro\\_script.zip](http://www.variosystems.ch/images/Downloads/Cadence_Allegro_script.zip)

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_NAME!NET_NAME!PIN_X!PIN_Y!START_LAYER_NAME!END_LAYER_NAME!GRAPHIC_DATA_NAME!G
RAPHIC_DATA_1!GRAPHIC_DATA_2!GRAPHIC_DATA_3!GRAPHIC_DATA_4!GRAPHIC_DATA_5!GRAPHIC_
DATA_6!GRAPHIC_DATA_7!GRAPHIC_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.**

---

### 2 EE Designer III ASCII File

**Common File Extension:** .ala

---

### 3 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
```

---

### 4 Incases TL CAD File

**Common File Extension:** .tl

## 5 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
```

---

## 6 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

---

## 7 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
```

---

## 8 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:**

\*.ca

**d File Header:** See GenCAD section

## 9 13.5

- 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
```

---

## 10 PANTHEON PDB File

**Common File Extension:** none

---

## 11 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}")
)
  
```

## 12 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=100000.000|ELECTRICALGRIDRANGE=0.5mil|ELECTRICALGRIDENABLED=TRUE|
SNAPGRIDSIZE=5000.000000|SNAPGRIDSIZEEX=5000.000000|SNAPGRIDSIZEY=5000.000000|TRACK
GRIDSIZE=50000.000000|VIAGRIDSIZE=200000.000000|COMPONENTGRIDSIZE=5000.000000|COMP
ONENTGRIDSIZEEX=5000.000000|COMPONENTGRIDSIZEY=5000.000000|CURRENTWORKINGLAYER=BOTT
OM|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)
  
```

## 13 SCI Cards Neutral File

**Extraction Procedure:**

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

**Common File Extension:** \*.cii

## 14 Supermax CAD

**Common File Extension:** \*.ipl

---

## 15 Tango for DOS

See PCAD .pcb export.

---

## 16 Ultiboard

### Extraction Procedure:

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

**File Header:** See GenCAD

---

## 17 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.

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

---

## 18 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))
```