



# Aegis CAD Imports

## Supported CAD formats and options



---

## Contents

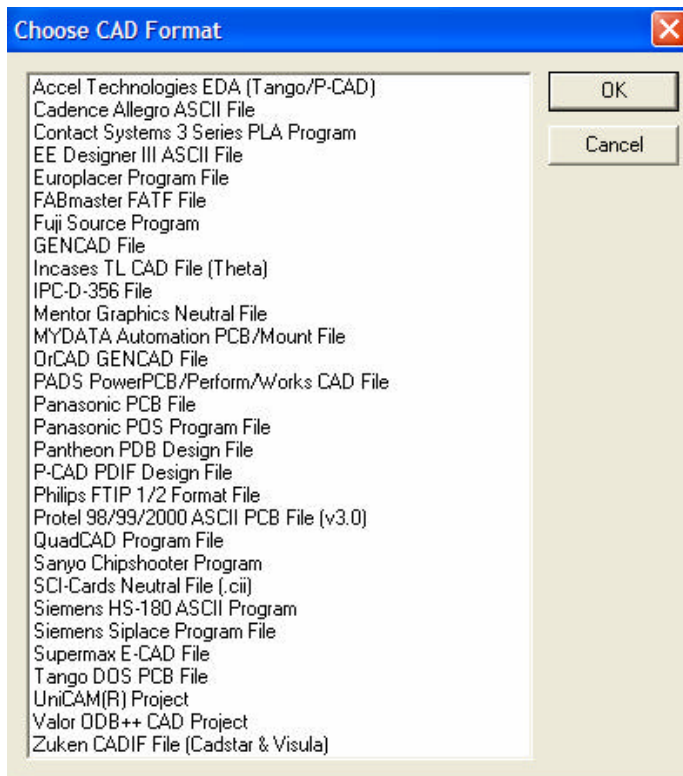
CAD Imports Overview .....	1
Supported CAD Formats .....	2
Source Program Imports .....	3
Supported Machine Program Formats .....	3
CAD Import Options .....	4
Using CAD Import Options .....	5
Fuji Source Program Options .....	5
GENCAD/ OrCAD GENCAD File .....	6
Mentor Graphics Neutral File .....	7
PADS PowerPCB/Perform/Works CAD File .....	8
Protel 98/99/2000 CAD File (v3.0 or above) .....	9
Supermax CAD File .....	10
Zuken CADIF File .....	11

---

## CAD Imports Overview

CircuitCAM supports a variety of CAD imports. Many of these CAD formats have additional import options that are controlled by the user. The following document describes Aegis supported CAD formats and any available options.

In CircuitCAM, select Import/CAD File from the Main Menu Bar.



---

## Supported CAD Formats

Listed below are the Aegis supported CAD imports and common file extensions.

<b><u>CAD Format</u></b>	<b><u>Common File Extension</u></b>
Accel Technologies EDA (Tango/P-CAD)	.pcb
Cadence Allegro ASCII File	.cad
EE Designer ASCII File	.ala
FABmaster FATF File	.asc
GENCAD File	.cad
Incases TL CAD File (Theta)	.tl
IPC-D-356 File	.net or .356
Mentor Graphics Neutral File	*.neu
OrCAD GENCAD File	.cad
PADS PowerPCB/Perform/Works CAD File	.asc
Pantheon PDB Design File	none
P-CAD PDIF Design File	.pcb, *.pdf
Protel 98/99/2000 ASCII PCB File (v3.0)	.pcb
SCI-Cards Neutral File (.cii)	.cii
Supermax E-CAD File	.ipl
Tango DOS PCB File	.pcb
UniCAM® Project	.pdw
Valor ODB++ CAD Project	*.tar or .tgz
Zuken CADIF File (Cadstar & Visula)	.paf

---

## Source Program Imports

CircuitCAM also is capable of importing several machine source program formats. These source program files may be generated in CircuitCAM or from any of the associated machines.

### Supported Machine Program Formats

Contact Systems 3 Series PLA Program (.PLA)

Eurolacer Program File (.TXT)

Fuji Source Program (.PRG)

MyData Automation PCB/Mount File (.PCB)

Panasonic PCB File (.PCB)

Panasonic POS Program File (.POS)

Phillips FTIP 1/2 Format File (.PRG)

QuadCAD Program File (.CAD)

Siemens Siplace Program File (.QD)

Siemens HS-180 ASCII Program File (.DAT)

Sanyo Chipshooter Program (.NCZ)

---

## CAD Import Options

The following CAD imports have no additional import options and can be directly imported into CircuitCAM.

Accel Technologies EDA (Tango/P-CAD)

Cadence Allegro ASCII File

FABmaster FATF File

IPC-D-356 File

Pantheon PDB Design File

SCI-Cards Neutral File (.cii)

Tango DOS PCB File

Valor ODB++ CAD Project

Eurolacer Program File

MyData Automation PCB/Mount File

Panasonic PCB File

Panasonic POS Program File

Phillips FTIP ½ Format File

QuadCAD Program File

Sanyo Chipshooter Program

Siemens HS-180 ASCII Program File

Siemens Siplace Program File

P-CAD PDIF Design File

EE Designer ASCII File

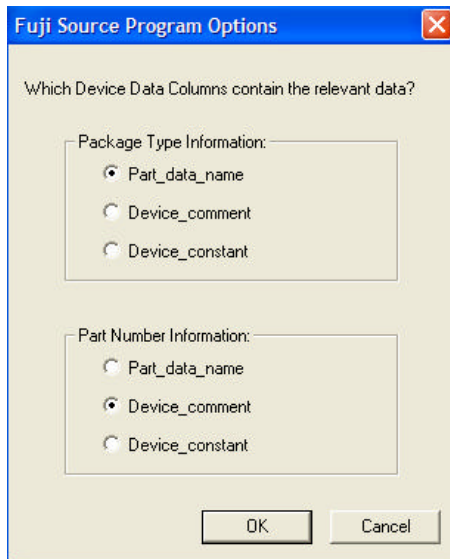
Incases TL CAD File (Theta)

## Using CAD Import Options

The following section describes CAD formats that have additional import options.

### Fuji Source Program Options

When importing a Fuji Source Program, users must define which columns of data are relevant to package and part number information.

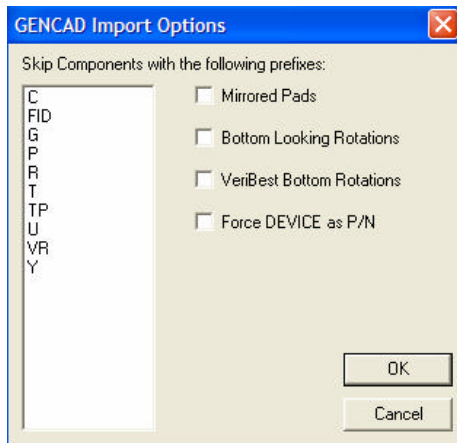


Package Type Information – Define which field contains package information.

Part Number Information – Define which field contains part number information.

- Part Data Name – Commonly refers to the Part Data File Name as supplied in the Setup Machine Dialog within CircuitCAM.
- Device Comment – Commonly refers to the Comment Field as supplied in the Setup Machine Dialog within CircuitCAM.
- Device Constant – This field may also contain part number or package information depending on how the program was generated.

## GENCAD/ OrCAD GENCAD File



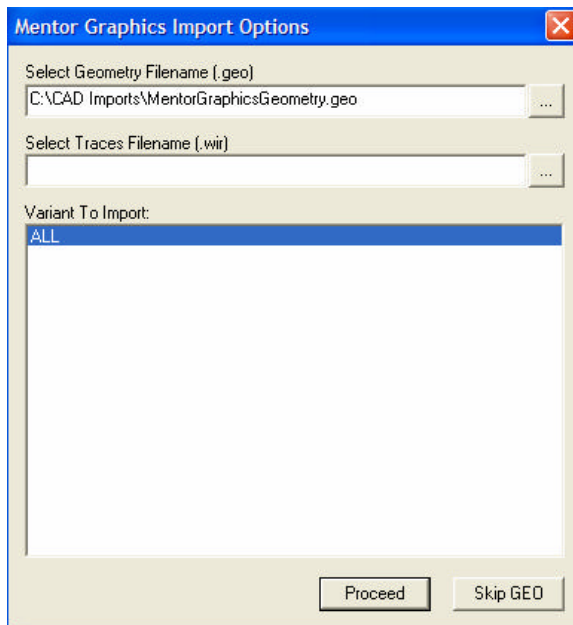
Skip Components with the following prefixes – Users will have the ability to select any prefix supplied by the designer. This will exclude any component with that prefix from the import. This is usually helpful when vias or test points are present.

Force DEVICE as P/N – If this option is checked, the part number information for the component will be imported directly from the "DEVICE" Field.

Due to the different interpretations of the GENCAD format from different CAD systems, Mirrored Pads, Bottom Locking Rotations and VeriBest Bottom Rotations are used to adjust how the cad file is imported and displayed. These settings are adjusted on a per CAD File basis.



## Mentor Graphics Neutral File

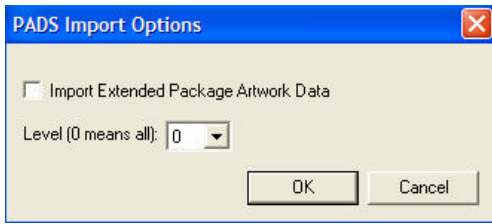


Select Geometry File (.geo) – This file is meant to enhance the package artwork as well as the board image displayed in the imported neutral file.

Select Traces Filename (.wir) – This option is currently inactive.

Variant To Import – If the designer has included different revisions within the CAD file, the user will have the option to select which revision or "Variant" is to be imported. Different variants may include different components, board frames, panel frames, root image, panelized assemblies, etc...

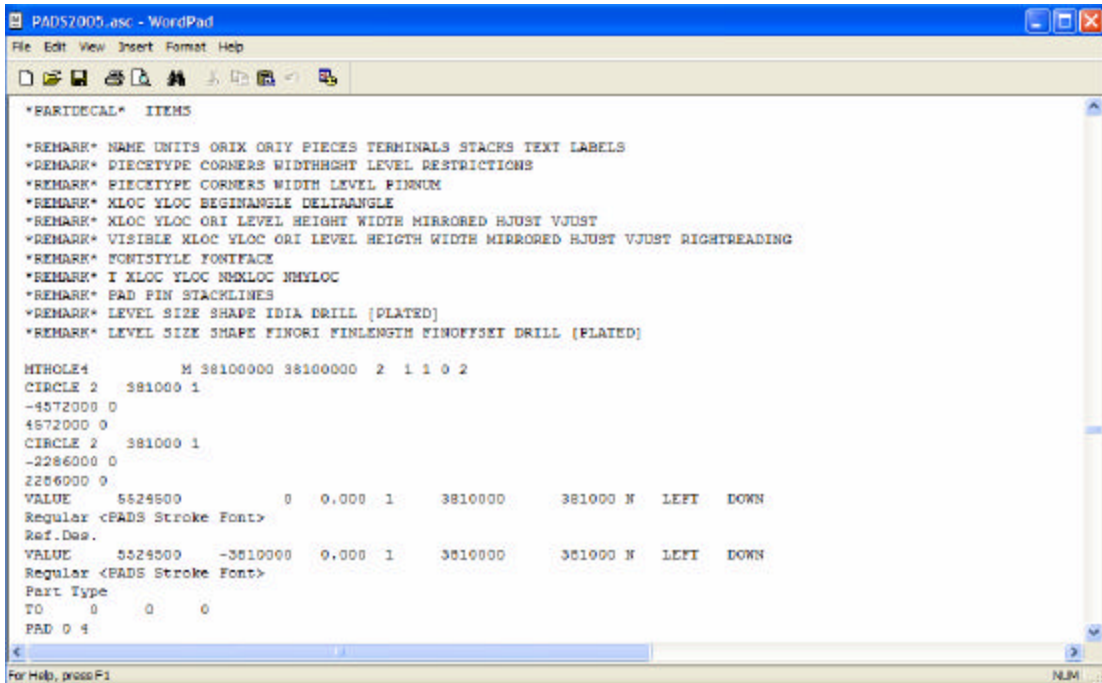
PADS PowerPCB/Perform/Works CAD File



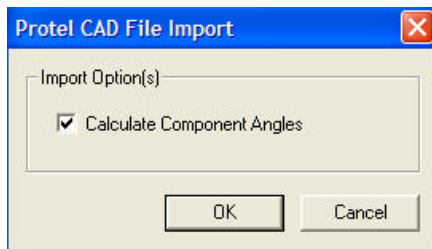
Import Extended Package Artwork Data – The PADS Importer will import the text based package artwork information from Part Decal section of the PADS file.

Level (0 means all) – This option includes the ability to import or not import different levels of the CAD file. Levels are defined by the section headers within the CAD file.

Example: PARTDECAL Section of a PADS File



Protel 98/99/2000 CAD File (v3.0 or above)



Calculate Component Angles – There are two rotations present in each Protel CAD File. One is the Package Rotation and the other is Component Rotation. If this option is selected, CircuitCAM will use the Component Rotation as defined in the CAD File. If it is not selected, CircuitCAM will use the Package Rotation as defined in the CAD File. Generally, this option should be selected. Below are examples of the same part with two different rotations within one CAD File.

Component Rotation:

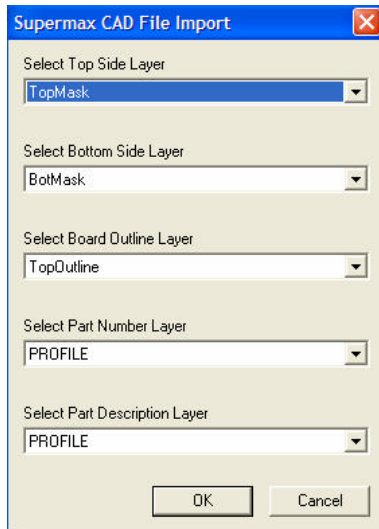
```
| RECORD=Text | COMPONENT=86 | SELECTION=FALSE | LAYER=TOPOVERLAY | LOCKED=FALSE | POLYGONOUTLINE=FALSE | USERROUTED=TRUE | X=4325mil | Y=4210mil | HEIGHT=45mil | FONT=DEFAULT | ROTATION=0.000 | MIRROR=FALSE | TEXT=U3 | WIDTH=6mil | DESIGNATOR=True
```

Package Rotation:

```
| RECORD=Component | ID=86 | SELECTION=FALSE | LAYER=TOP | LOCKED=FALSE | POLYGONOUTLINE=FALSE | USERROUTED=TRUE | PRIMITIVELOCK=TRUE | X=4380mil | Y=3960mil | PATTERN=LCC32_S | NAMEON=TRUE | COMMENTON=FALSE | GROUPNUM=0 | COUNT=0 | ROTATION=180.000 | HEIGHT=0mil | NAMEAUTOPOSITION=0 | COMMENTAUTOPOSITION=0 | UNIONINDEX=0
```

## Supermax CAD File

To import Supermax CAD Files, the user will need to define which layers are to be used for top or bottom side of the board. Users will also have to define part number and package layers as well as outline layers.



Select Top Side Layer – Select which layer is to be used as the Top Side Layer. This layer will contain components that are placed on the top side of the board.

Select Bottom Side Layer - Select which layer is to be used as the Bottom Side Layer. This layer will contain components that are placed on the bottom side of the board.

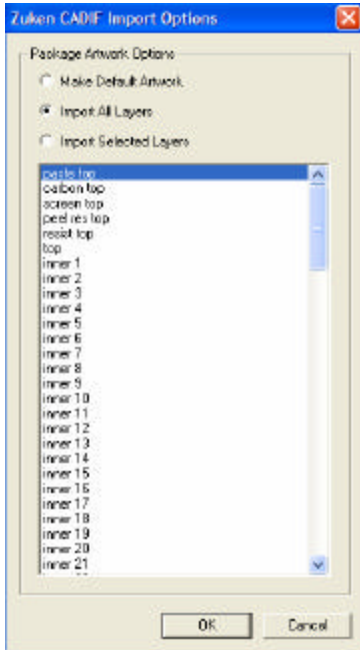
Select Board Outline Layer – This layer contains board outline information such as panel or image frames.

Select Part Number Layer– Defines the field or column that contains part number information.

Select Part Description Layer – Defines the field or column that contains the part description/package information.

## Zuken CADIF File

Users have several options for importing Zuken CADIF Files. Users can choose to import all layers or selected layers. If the CAD File does not contain package artworks, the user can use CircuitCAM's default package shapes and apply them to the components.



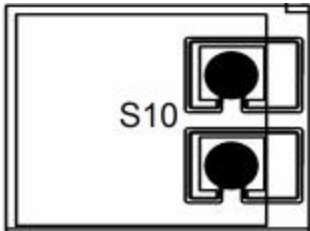
Make Default Artwork – This option will apply CircuitCAM's default package shapes to components.

Import All Layers – Imports all CAD File layers.

Import Selected Layers – Users define which layers are to be imported.

Examples:

1A. Component when imported with the CAD supplied artwork



1B. Component when imported with CircuitCAM's default artwork

