

Components in CAM Software

Jun 2021

Introduction

Component information for PentaLogix CAM data can come from several sources:

1. From importing **ODB++** data. When using the new version of ODB++ import, component information is imported as component elements and stored on separate layers. The layer types are: CMU (component upper) and CML (component lower).
2. Component elements can be **created** in the CAM software from CAM elements (pads, traces, polygons).
3. Using the **EMS Magician** module (included with CAMMaster or added to the other products) one can parse pick and place files (PnP) and bill of materials files (BOM) to feed into the CAM software.
4. From importing netlist data (**IPC-356**). If component data is present in the netlist data, it is attached to the CAM elements (pads) as custom properties.

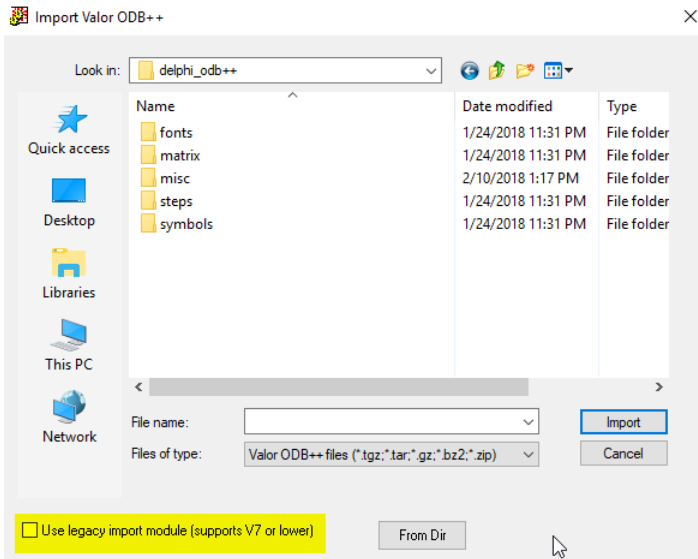
The different versions of CAM software expose different levels of functionality related to components.

All versions allow importing and viewing, including the free ViewMate. Selecting by package type is implemented in ViewMate Deluxe and up. All other functionality is available in ViewMate Pro, CAMMaster Designer and CAMMaster.

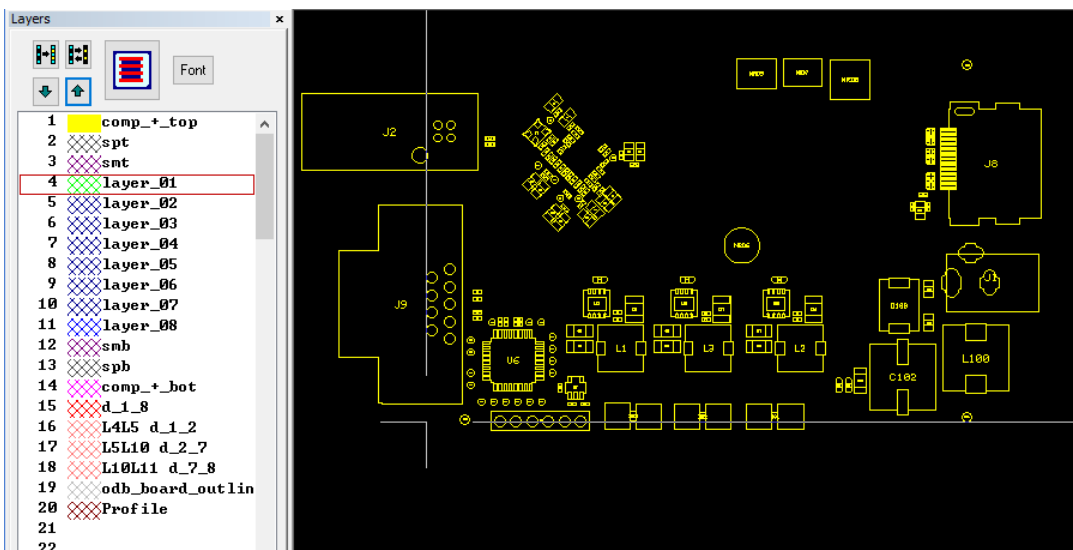
An important feature is **validating** component info versus the copper data. The main capability here is to make sure that component footprints from pick and place files match the geometry of the copper data before starting the assembly process. Sometimes wrong part numbers appear in PnP files resulting in problems after the assembly process has started.

Importing From ODB++

When importing an ODB file or folder, make sure that “Use legacy import...” is not checked. The legacy import (older version of the import code) does not handle components.



An example of what is seen after the import is shown below:

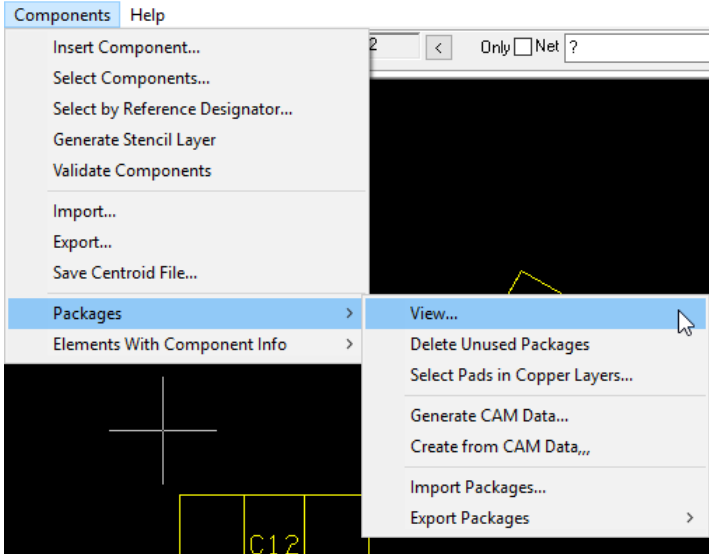


Note that not all ODB++ files contain component info.

Packages

Viewing the Available Packages

To bring up the packages dialog use this:



Components in CAM

You will see a dialog with all the package info that exists in the loaded data:

The 'Packages' dialog box displays a table of package information. The table has the following columns: #, Name, Part#, Pins, Size, and Pitch. The first row is selected, showing package #1 with Name 'fnls_acs02229a1_rn', Part# empty, Pins 33, Size 5.3000 x 5.3000 mm, and Pitch 0.5000 mm.

#	Name	Part#	Pins	Size	Pitch
1	fnls_acs02229a1_rn		33	5.3000 x 5.3000 mm	0.5000 mm
2	fnls_acs02229a1_no		33	5.3000 x 5.3000 mm	0.5000 mm
3	fnls_act00896a1_fb		9	3.6000 x 3.1000 mm	0.6500 mm
4	fnls_act00896a1_fc		9	3.6000 x 3.1000 mm	
5	fnls_act02233a1_el		49	7.6000 x 7.6000 mm	0.5000 mm
6	fnls_act02233a1_em		49	7.6000 x 7.6000 mm	0.5000 mm
7	ihdn_ebu01649a0		6	7.9000 x 9.4500 mm	0.8000 mm
8	ihdp_ddx00567a0		6	7.9000 x 9.4500 mm	0.8000 mm
9	ihdp_ddx00567a0_at		6	7.9000 x 9.4500 mm	0.8000 mm
10	leds_afi00291a0		2	9.0000 x 4.0100 mm	1.9500 mm
11	mofp_afm00015a0		2	8.7500 x 6.2200 mm	7.1700 mm
12	mofs_afa00179a0		2	7.8000 x 7.2000 mm	6.3500 mm
13	mofs_afa00183a0		2	11.0000 x 10.4000 mm	8.3000 mm
14	mofs_afa00292a0		2	2.9000 x 1.5000 mm	1.9500 mm
15	monp_agd06030a0		2	2.3000 x 0.9500 mm	1.4500 mm
16	monp_age08050a0		2	2.8000 x 1.4500 mm	1.8500 mm
17	monp_ahw08050a0		2	3.2000 x 1.4500 mm	2.0500 mm
18	monp_ahw12060a0		2	4.4000 x 1.8000 mm	3.2000 mm
19	monp_ahw12100a0		2	4.6000 x 2.8000 mm	3.3000 mm
20	monw_fba06035a0		2	3.1500 x 0.9500 mm	1.9500 mm
21	monw_fbc08056a0		2	4.6000 x 1.4500 mm	2.8000 mm
22	monw_fbe12066a0		2	6.0000 x 1.8000 mm	3.9000 mm
23	mofa_afa00186a0		100	16.7000 x 16.7000 mm	0.5000 mm

Below the table is a preview window showing a footprint of a package. The footprint is a square with pins on all four sides. A mouse cursor is pointing at the footprint. To the right of the preview is a search box labeled 'Search for name:' and a 'Delete Unused' button. Below that is a 'View' section with radio buttons for 'Footprint' (selected) and 'Stencil'. At the bottom are 'OK' and 'Cancel' buttons.

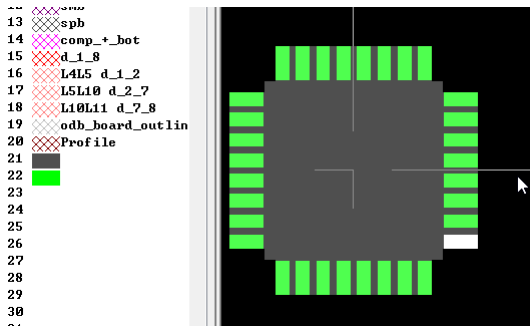
If a package was selected in the CAM editor, it will appear as the selected row.

Generate CAM Data from Package Data

This is useful for editing the outline / pins of a package (footprint) so it can be edited or used as a starting point for making a different package template.

Use “**Components>Packages>Generate CAM Data**”.

After picking the package from the “Packages” dialog, new data will be generated in the first empty layer found (two or three consecutive empty layers are needed):



The first layer (gray) will contain one polygon – the package outline

The second layer (green) will contain the pins as polygons, with pin 1 selected.

The third layer (optional) will have the stencil outlines as polygons. This will only appear for packages that contain stencil data.

The data is generated so that the package center is at (0,0).

Create a New Package from CAM Data

This is the reverse of the above operation. Useful for creating new packages to be used as components.

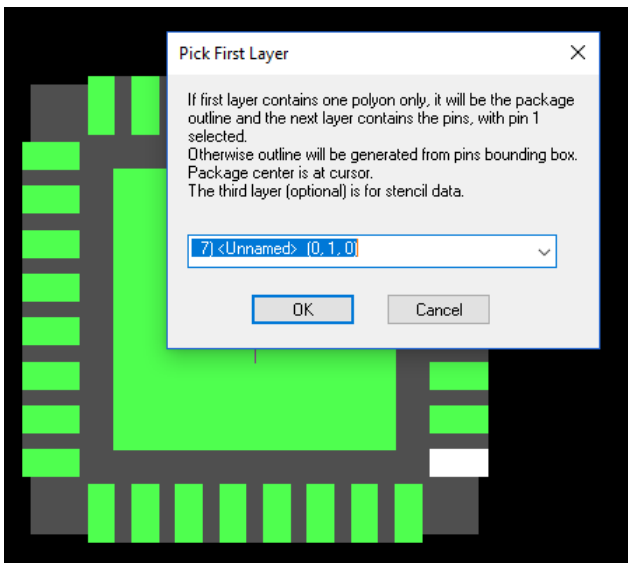
One can start from:

- Data generated as shown in the previous section and then edited
- Pad data from a copper layer
- Completely new data created using the CAM editor (pads, traces, polygons)

The data needs to be the only data on one, two or three consecutive layers.

Use the command “**Components > Packages > Create From CAM Data**”

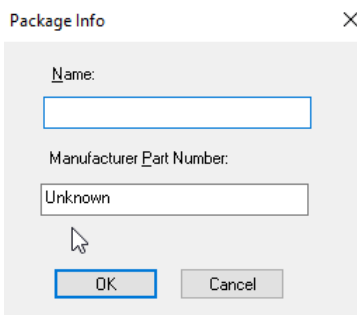
You will be prompted with a dialog to pick the layer where the data resides:



The rules that determine how the data is interpreted are below:

- The package center will be at the current cursor position. Because of this, make sure to position the cursor as desired then detach the cursor (Space bar) before executing this operation.
- The data should be on one, two or three layers starting at the picked layer.
 - If the first layer contains only one polygon, it is assumed to be the outline and the second layer must contain the pins data.
 - If the first layer contains more than an element, it is assumed to contain the pin data (as polygons, pads or traces) and the outline will be automatically generated from the bounding box of the pin data.
 - If the next layer following the pins layer has any data, it will be assumed to contain the stencil outlines for this package (as polygons, pads or traces).
- One can specify pin number 1 by selecting one element in the pin data layer, although this is not required.

You will then be prompted for the package name and part number:



The name cannot be left blank. The part number, if known, should be added here (will be useful for generating BOM files), but it can also be edited later, if not known at this time.

Importing and Exporting Package Data

Use “**Components > Packages > Import Packages**” and “**Components > Packages > Export Packages**”. This reads in or writes package libraries (from the currently loaded packages) that can be reused later.

Several formats are supported:

1. Proprietary binary format, “PentaLogix Packages”. Extension “.plxpkg” is mandatory.
2. ODB++ EDA format. This is the same format as the file in the “eda” folder of ODB++ data. This is a text file that is human readable.
3. KiCad footprints. These can only be imported, export is not supported.
Many different footprints are available at <https://kicad.github.io/footprints/>. Download any that you need and import.

Selecting Pads in Copper Layers

Use “**Components > Packages > Selects Pads in Copper Layers**”.

After you pick a package, pads in the copper layers (CPU and CPL) will be selected. The selected pads have centers inside the pins of components of the given package on the corresponding components layer (CMU or CML).

Stencil Outlines

This part of the package data is optional.

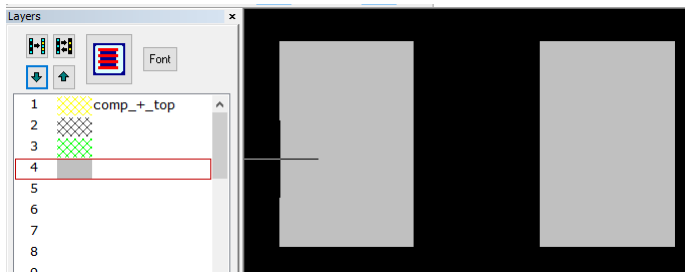
Below are the steps that can be used to create the stencil outlines for packages that do not have them.

First you need to generate CAM Data for the package. See “Create a New Package from CAM Data” above.

Now you can use the pins layer data to generate the stencil data.

In the layers toolbar right-click on the layer with the pins data and choose “Edit/Replicate”. Replicate the data to the next layers (one layer number higher).

For this example we are using a package with two rectangular pins:



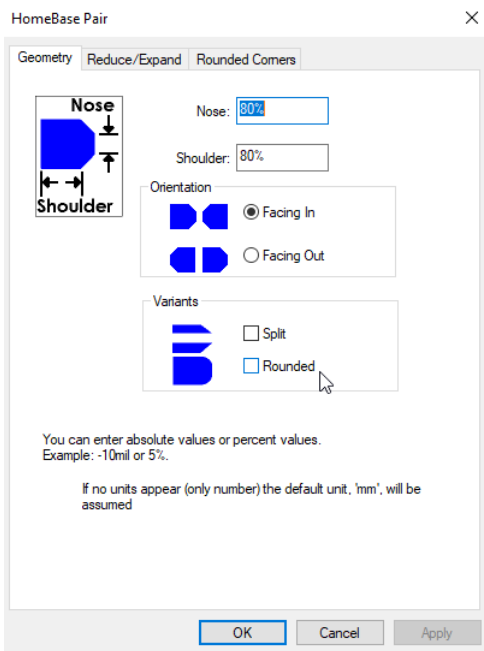
Now we need to use the stencil module from the software to generate the stencil outlines. The stencil module works on pads and here we have polygons, so the first step is to convert these polygons to pads.

1. Select both polygons
2. Do “**Tools > Convert Drawn Pads > Global**” and use the “Standalone pads” option. Most other default settings will do.

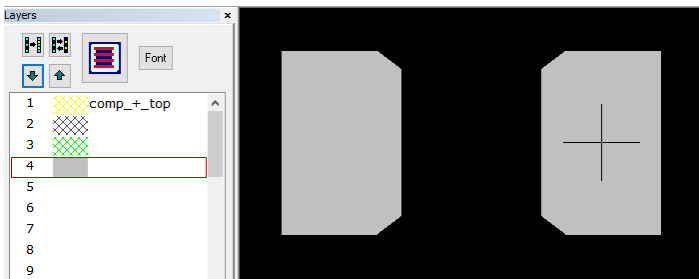
This will convert the two polygons to two rectangular pads.

Now we can use the stencil module to generate the desired stencil outlines. Let us assume we want homebase shapes.

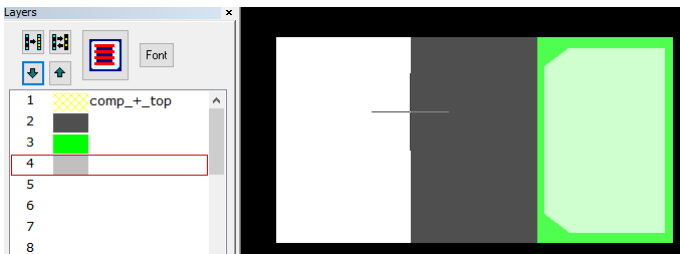
Select the two pads and do **“Tools > Stencil Aperture > Homebase”**. Assume the following parameters:



The data is changed as shown below:



Now make layer 2, 3 and 4 visible again and select the outline for pin 1 on layer 3:

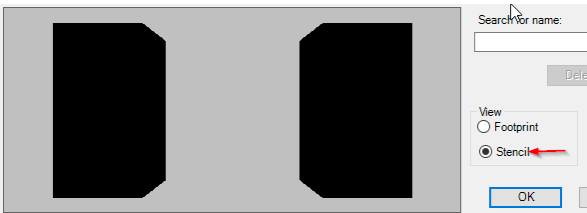
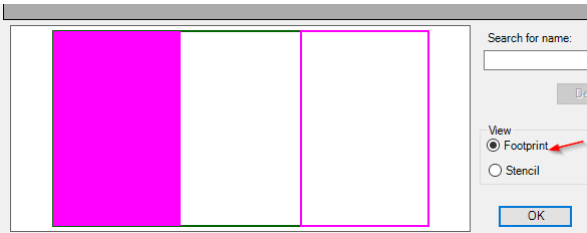


Last step is **“Components > Packages > Create from CAM Data”**. Pick layer 2 as the first layer.

You will next be prompted for the package name. You can give it a new name or use the same name to update the existing package with the added stencil info.

Components in CAM

Once stencil info has been added to a package it can be viewing in the package view dialog.



Components

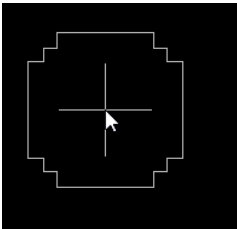
Adding New Components

Once one has package data loaded one can also insert/add new components.

This is useful when manually creating components for a Gerber based design.

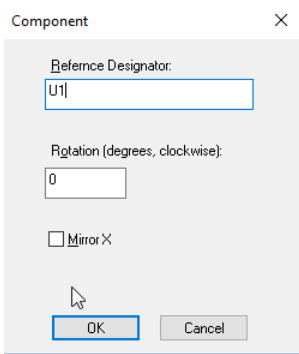
Use “**Components > Insert Component**”. You will see the Packages dialog to pick a package. This assumes that you have package data loaded (from importing from a library or created in this session).

After picking a package the view will show a movable outline:



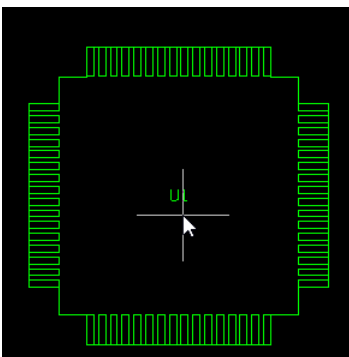
Move the cursor (with the dragged outline) to the desired location and click the mouse or press Enter.

You will see this dialog:



Enter the desired info and click OK.

The component will be inserted in the current active layer:



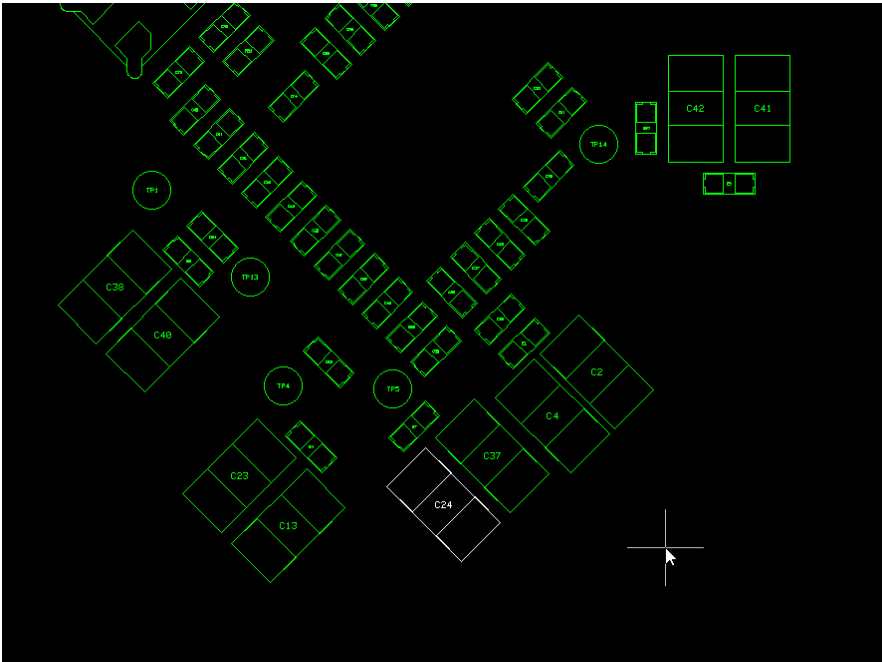
In this way, if one does not have component info to begin with (only Gerber data) one can generate component layers to match the data.

Selecting Components

Use “**Components > Select Components**”. This selects all components of a chosen package.

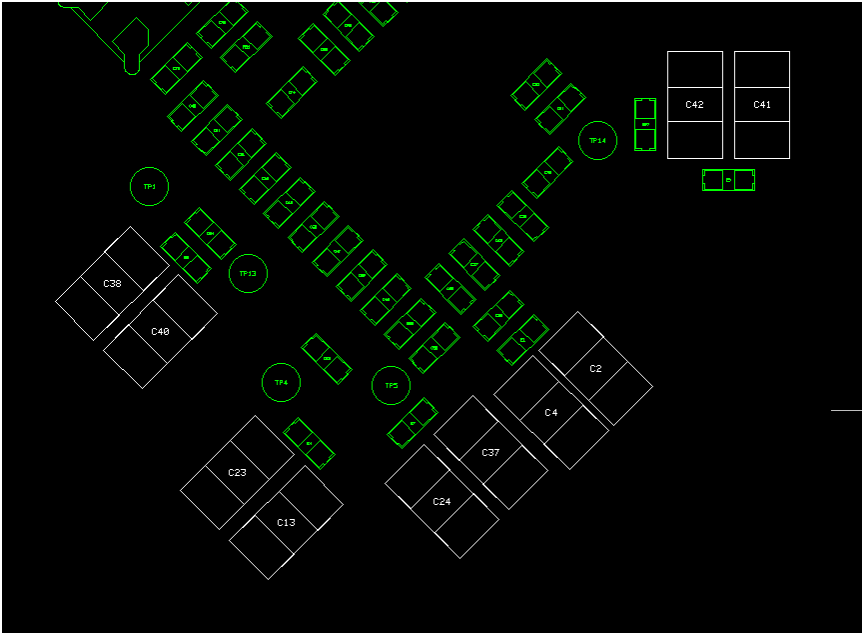
You will be prompted to pick a package. All components of that package will be selected in the visible layers.

So, to select all components with the same package as one that is of interest, first select a component by double-click.



Do “Select Components”. The package of the previously selected one will show. Click OK.

All components with this package will now be selected:



Importing and Exporting Components

This can be done with “**Components > Import**” and “**Components > Export**”.
Data is in *PentaLogix Component File Format* described later in this document.

Saving a Centroid File

This can be done with “**Components > Save Centroid File**”.
It will save a .csv file and you will be prompted for the separator character.
The units are the current units and the angles are counterclockwise.
The columns are:

- Reference Designator
- Center X
- Center Y
- Angle
- Side

Note: if you have special requirements for the format of the centroid file, you can contact PentaLogix and we are open to adding new formats.

Validation

Command is “**Components > Validate Components**”.

Component data is validated vs the corresponding copper layers and the board outline.

CMU <-> CPU

CMU <-> BOL

CML <-> CPL

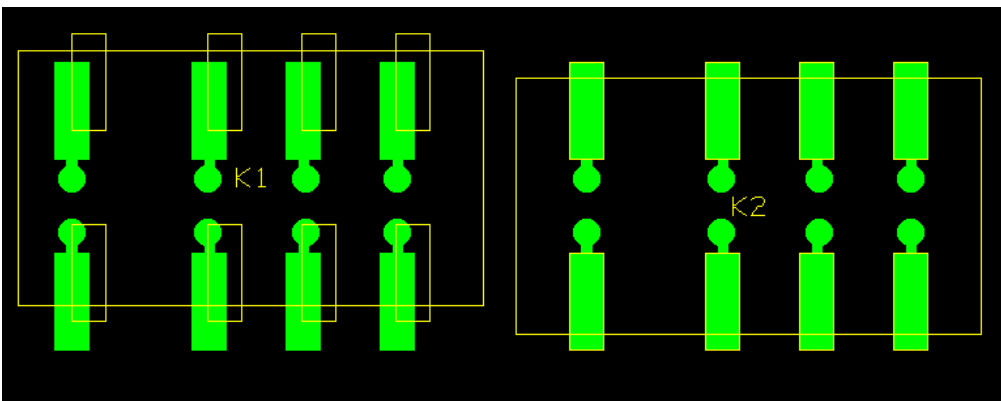
CML <-> BOL

Only layers that are visible will be checked. So, for instance, to disable checking against the BOL layer, turn that layer off.

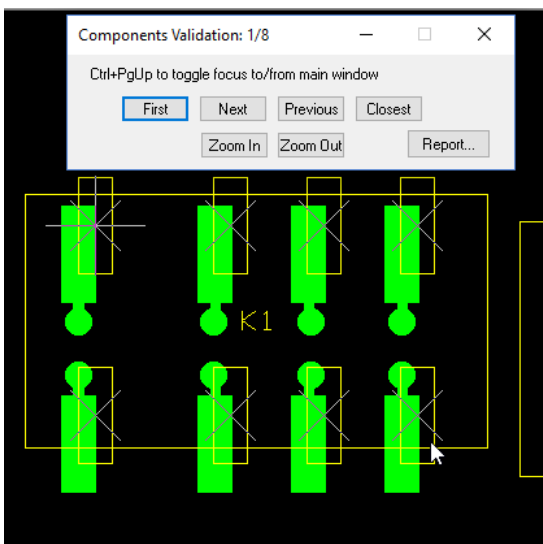
When checking against copper layers, pins in the component need to match copper (usually pads) in the copper layers. This will catch components that have a different footprint than the copper and also components that are displaced or rotated the wrong way.

When checking against the board outline, minimum distance of the component to the board outline is checked. The minimum allowed distance is 120 mils or 3 mm.

A trivial example below shows a misplaced component:



After running the validation markers are shown at the places where the pins (yellow) do not match the copper (green) with a navigation dialog:

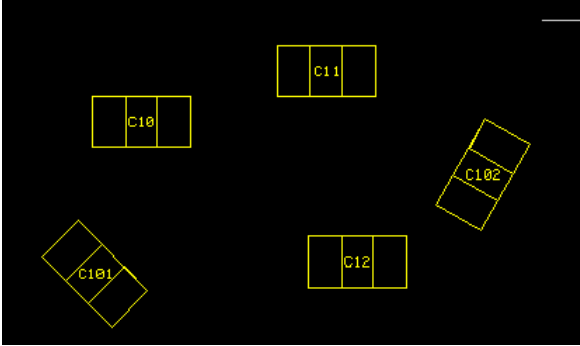


Generate Stencil Layer

"Components > Generate Stencil Layer".

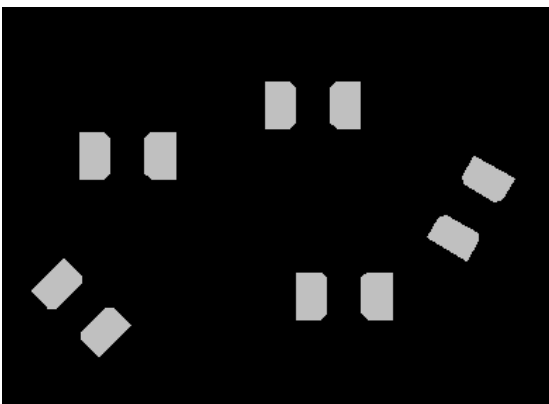
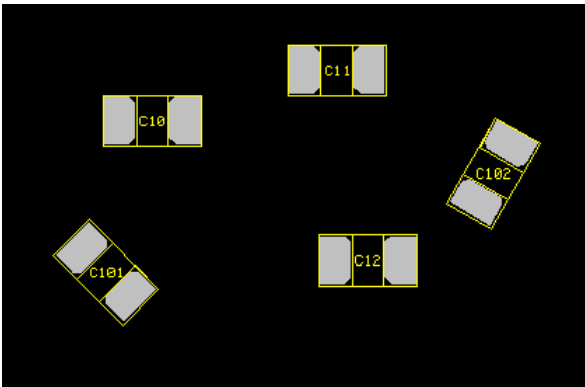
This works only for packages that have stencil info (see “Stencil Outlines” section above). It is very powerful because stencils at any rotation can easily be generated.

Let us take the following simple example:



All five components are based on the same package and the package has stencil info. Now do: **“Components > Generate Stencil Layer”**.

The stencil elements will be inserted on the first empty layer found:



Two layers may be generated if there were two component layers visible (upper and lower) and they had stencil info.

Converting Pad Data to Components

“**Components > Convert Footprints to Components**” will find groups of pads and convert them to components.

There is a separate app note describing this called “Convert Pads To Components”.

File Formats

There are two types of component files that the CAM software and EMS Magician can export and import.

1. A components data file, similar to a pick and place file but standardized for PentaLogix use. The extension of this file is “.plxnpn”. It is human readable and can be opened in Excel as well. Format is given below.
EMS Magician can generate this file from a compilation of items from BOM and the PnP files (many commonly available formats supported).
2. A package library file that contains the data for the packages used (this is also called “Attached Footprint Library” in EMS Magician). The extension of this file is “.plxpkg”. In essence, this is a custom footprint library file that includes the footprint info of all of the parts that are part of the current design.

PentaLogix Component File Format

It is a tab-delimited ASCII file that is human readable. Mandatory extension is “.plxnpn”

The file contains eight mandatory columns and two optional columns. A sample of the beginning of a Footprint Data file (including header names) is shown below:

	A	B	C	D	E	F	G	H	I	J	
1	REFDES	MFGPN	FOOTPRINT	LOCATIONX	LOCATIONY	SIDE	ORIENTATION	DNP	SIZEX	SIZEY	
2	U3	CD40107BM96	SOIC-8	0.63673	1.18059	Top	90	No	0.27284	0.17283	
3	U4	CD40107BM96	SOIC-8	1.75674	0.85559	Top	90	No	0.27284	0.17283	
4	R2	CRCW12063R90FKEA		1206	0.475	2.18504	Top	90	No	0.05512	0.17716
5	C2	TMK316B7106KL-TD		1206	0.74	1.95496	Top	90	No	0.05512	0.17716
6	R3	CRCW12063R90FKEA		1206	0.575	2.18504	Top	90	No	0.05512	0.17716
7	C9	TMK316B7106KL-TD		1206	1.545	1.86996	Top	90	No	0.05512	0.17716
8	C10	TMK316B7106KL-TD		1206	1.645	1.86996	Top	90	No	0.05512	0.17716

The end of the file contains three additional rows:

215	Units	inch	
216	CultureName	en-US	
217	AttachedLibraryName	32881.plxpkg	

The value for “AttachedLibraryName” is the name of the Package Library File. It can be a full path or just a filename (in which case it is found in the same folder as this file). The value can also be left empty, in which case it is assume that the CAM software already has the data for the packages needed by the components.

A description of each column follows:

Column Header	Description	Legal Values	Example
REFDES	Reference Designator	String	U3, R10A, J2_B
MFGPN	Manufacturer Part Number	String	CRCW1206XXX
FOOTPRINT	Footprint Name – This will link to a footprint structure (package name) in the attached library file	String	SOIC-8
LOCATIONX	The X axis location of the center of the footprint	Double	0.467595

Components in CAM

		Values less than 1 must have leading 0.	
LOCATIONY	The Y axis location of the center of the footprint	Double Values less than 1 must have leading 0.	12.365894
SIDE	Determines whether the component is on top or bottom of board	Top Bottom	Top
ORIENTATION	Angle of the footprint relative to the defined footprint orientation. Footprints are always created at 0 degrees.	Value is in degrees. Two significant digits. Counterclockwise rotation is positive.	90
DNP	Do Not Populate. If set to yes, then part is not to be populated.	Yes No	Yes

In addition, there are two optional columns called "SIZEX" and "SIZEY" for EMS Magician internal use.

PentaLogix Package Library File Format

This is a proprietary binary file that is not human readable. Mandatory extension is ".plxpkg".

It contains package information for the components.

Component Info Attached to Copper Pads

This feature was available in previous versions of the CAM software as well (pre V11.14).

The component information is added as custom properties after importing an IPC-356 netlist.

Overview

The custom property names are:

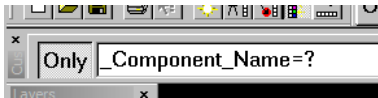
- `_Component_Name`: for the name of the component
- `_Component_Type`: for the type of the component.
- `_Component_Pin`: for the name of the pin (usually a number, but can be alphanumeric)

Note that these properties start with an underscore, because they are system custom properties, with predefined meaning in the PentaLogix CAM software.

They can be inspected and used via the custom properties toolbar (“View/Toolbars/Custom Properties”). When this toolbar is visible, it will display the custom properties of the currently selected element or the currently inspected element (via “Go/Active Element/Closest” or Shift+A).

Selecting Components

When the “Only” button in the custom properties toolbar is checked, the property shown can be used as a criterion in the selection. The concepts involved in custom properties are explained in a separate document, but below is an example which will select all elements that have a component name:



Note that the “Only” button is pressed and the box contains “_Component_Name=?”.

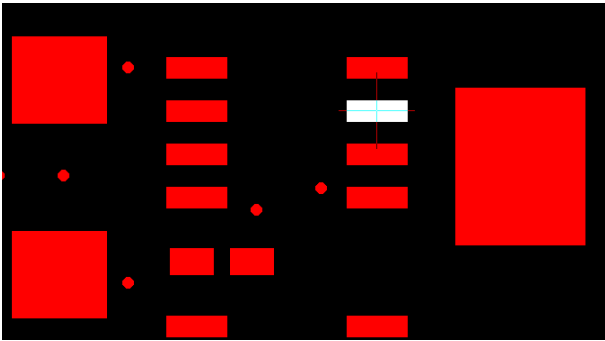
If at this point one does a global selection (“Select/Global/New” or V), all the elements that have a component name (any name, since “?” matches all names) will be selected.

There is also a separate command to select components: “Select/Components”. The dialog that comes up shows all component names (or types) in two panes.

The left pane, labeled “Visible” shows all components that are currently visible.

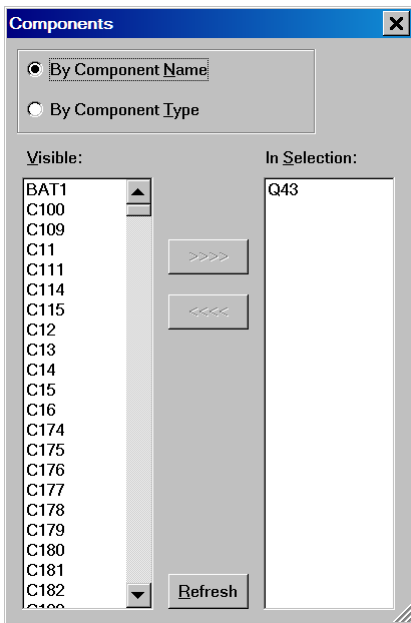
The right pane, labeled “In Selection” shows the components that have elements in the selection (if any). One can select and deselect elements of certain component names (or types) by picking the components in the panes and then clicking the buttons to transfer them from one pane to the other (the “>>>>” or “<<<<” buttons). This dialog will stay up until closed and one can have access to the main view at the same time. However, it does not refresh automatically when changes are done to the main view. If a refresh is needed, please click on the “Refresh” button.

A sample sequence of steps for using component names in selecting is shown below. In this sample file a pad has been selected, by double clicking on it.

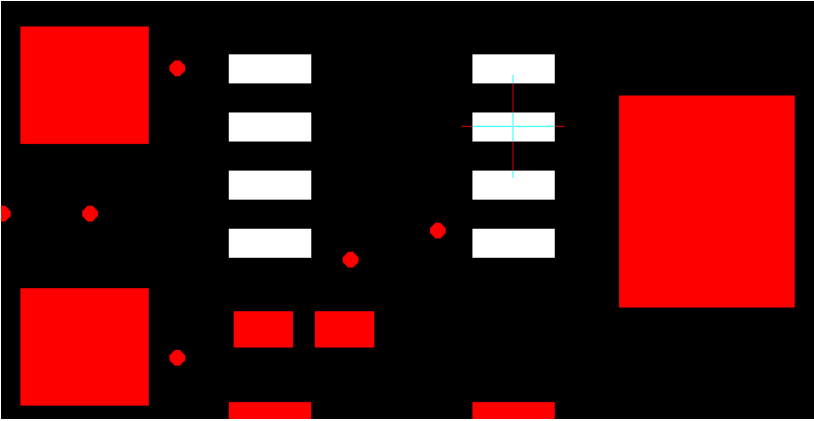


After “Select/Components” the following dialog show below comes up. It shows that the selected pad had component name “Q43”.

If we now want to select all elements with this component name, we can pick “Q43” from the “Visible” pane and then push “>>>>”.



Alternatively, we can pick “Q43” from the “In Selection” pane (because it is easier to find) and then transfer focus to the view (by clicking on it, or via Ctrl+PgUp) and do a global select (V). The selection is shown below, with all pads of this component selected.



Note that one could have used the selection frame to select these pads also. And finally, when doing a selection inside the “Select/Components” dialog, the “Only” buttons are also taken into consideration, so one can limit the selection to specific layers, D Codes, etc.