

CAMMaster Release Notes

This document covers CAMMaster, CAMMaster Designer and CAMMaster64. Not all new features are available in CAMMaster Designer.

New Features

11.24.50

- **Mirrored View**
“View > Mirrored on X” will toggle the mirrored view mode.
The loaded data will not mirrored only the view.
- **New EMS Magician**
V1.2.4

11.24.43

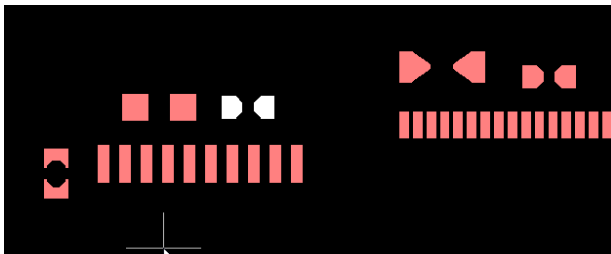
- **New EMS Magician**
V1.2.3

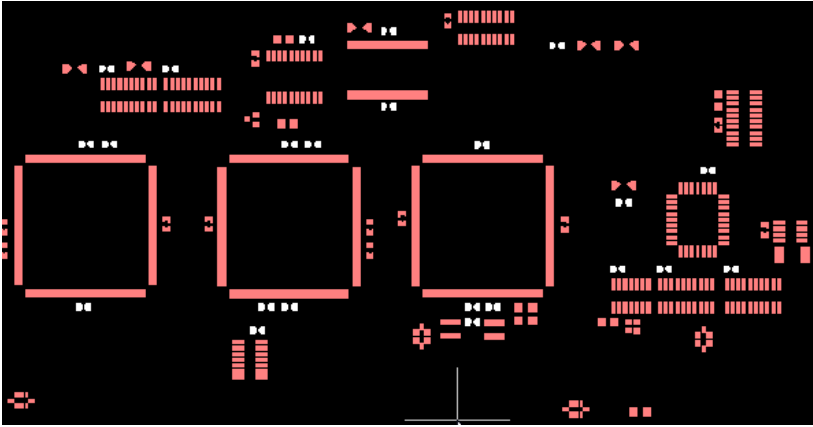
11.24.39

- **Export ODB++**
Now includes component info (from component type layers, CMU and CML).

11.24.36

- **New in the D Codes Setup Table (F5)**
 - Changing Shape of D Codes While Preserving Dimensions: when selecting more than one D Code and changing the shape, the dimensions of the D Codes selected will be preserved.
This is useful, as an example, when changing a number of Q shapes to R shapes.
 - A new command to change all diamond shapes to rotated squares, preserving geometry: “Operations > Convert D to S”.
- **Selecting Patterns**
“Select > Similar Pattern”. Will select (in the same layer) all elements that are similarly arranged as the selected elements. Both position and shape are taken into consideration.
This is most useful for groups of component landings (either pads or polygons), but work for any type of elements.





- **Calculating Solder Paste Volume and Weight**

“Tools > Stencil Apertures > Ratios / Volume > Paste Volume / Weight”. Computes the volume of the paste for the selected elements assuming the thickness given by “Stencil Thickness”. The solder paste density needs to be given as a parameter.

11.24.30

- **Copy Coordinates to Clipboard**

New item in the context menu (right-click). Allows the current coordinates to be copied into another document if needed (such as Excel).

- **Combine the Data from Several Layers into One**

In the Layers Setup dialog (F10), select several layers, then right click and pick “Combine Selected”. This will move all the data into the first layer in the selection. The remaining layers will be empty.

- **Support for Subscription Licenses**

11.24.27

- **BGA and QFN Stencils**

Pick “General Component” for a stencil creation.

When all pads are round (C) and the patterns is a matrix, it will be interpreted as BGA (Ball Grid Array) and the only option is to replace the rounds with a shape taken from another D Code.

When pads are arranged in 4 rows outside with a rectangular belly pad inside, it will be processed as QFN, with window splits for the belly pad.

For more info please see “Stencil_Apertures.pdf” in the “Documentation” folder.

- **Import ODB++**

Component info, if present, is also transferred to CPU/CPL pads.

Support for “oblong_ths” shape.

- **Import IPC-2581**

Support for slots (“SlotCavity”).

11.24.23

- **Export StL**

Export all layers in StereoLithography (StL) format into a folder.

Useful to make 3D views with the appropriate software.

- **View Drill Stack Up**
 “View > Info > Drill Stack Up”.
 Shows in a text dialog the drill layers stackup relative to the circuit layers. Layer types must be set.

11.24.20

- **View D Code and Tool Codes Used In Certain Layers**
 Two new commands “Setup > D Codes in Layers” and “Setup > Tool Codes in Layers”.
- **Exclude Import of ODB++ Component Layers**
 “File > Import > ODB++ Options > Exclude COMPONENT Type Layers”.
 Component layers render more slowly and, when not otherwise needed, one can exclude from import.
- **Components Geometry**
 “Components / Generate Geometry Layers”.
 Generates layers corresponding to existing component layers, but having only the geometry (as traces).

11.24.15

- **Support for Gerber Extended APERFUNCTION Directive**
 This directive associates a D Code (aperture) with a function.
 An example is: %TA.APERFUNCTION,ViaPad*%
 Support has been added for both import and export. The function names are in the Gerber spec.
 The D Codes Setup table (F5) has an additional column for this, called “Function”. A blank value means there is no function specified.
 Currently the function field is only used for import and export, no special meaning for CAM internal operations.
- **Insert Empty Layer**
 In the Layers Toolbar context menu a new command: “Insert Empty Layer” will insert an empty layer before the current one and shift the current and all following down to make room.
- **Selects Pads with VIA Drill Holes**
 “Select > Special > Pads With VIA Holes” will select all pads in the visible conductive layers that have a hole of “VIA” type.
- **Specifying Pin 1 for Components**
 “Components > Convert Footprints to Components” now allows for forcing Pin 1 to a particular pad.
 This can be done by placing a marker on the center of the pad that is desired to be marked as Pin 1.
 Use “View > Mark” for this (shortcut “M”). This toggles a marker.

11.24.11

- **D Code Usage Per Layer**
 The “Layers” command in the D Code table now shows layers used for individual D Codes in the group of selected D Codes.
- **D Code Info in CSV Format**
 In the D Code Setup table “Operations > Export as CSV File” will create a file that can be opened in Excel.

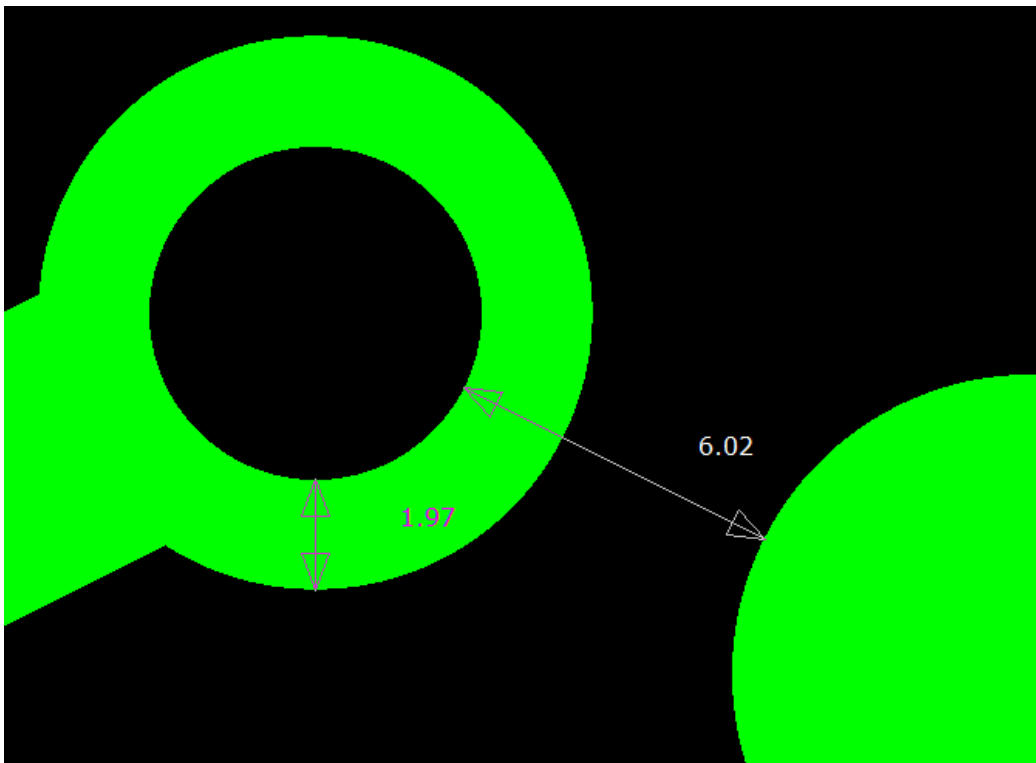
- **New Edge To Edge Distance Tool**
 “View > Distance > Edge To Edge Distance Tool” displays the distance between the edges of two elements.

11.24.3

- **Generate Soldermask**
 “Tools > Soldermask/paste > Generate Solderamask from PadMaster”.
 An option was added to add rounded corners to shapes that are rectangular.
- **Copy By Displacement**
 “Edit > Edit Selection > Copy By Displacement”.
 Works the same as “Move By Displacement” but does a copy instead of a move.

11.24.1

- **Change Coordinate of a Selected Pad or Linear Trace**
 “Edit > Edit Selection > Change Coordinates”.
- **Improved Drill DFM**
 Clearance checks are done in addition to annular ring checks.
 So, one can get two violations, like shown below:



- **New Release of EMS Magician (V1.2.1)**
 - Enable back button on every step without restrictions
 - Update Help Files
 - PCB Analyzer - Update "First" button behavior
 - Bug: Layers deleted when we switched to panelize module if the PCB Analyzer has not finalized all the

steps.

- Bug: On some rare occasions when the StartOver button was pressed there were temporary layers that remain active in viewmate
- Bug: Whenever button was click it load the Original temporary file saved by EMS instead of the loaded on viewmate.
- Bug: PCB Analyzer - Clip to board outline panel was not clearing after start over event
- Bug: PCB Analyzer - Identify layers was not updating the layers if DFA Settings changed after reordering layers
- Bug: If for some reason AppData files are removed the application crash on launch

11.22.53

- New Release of EMS Magician

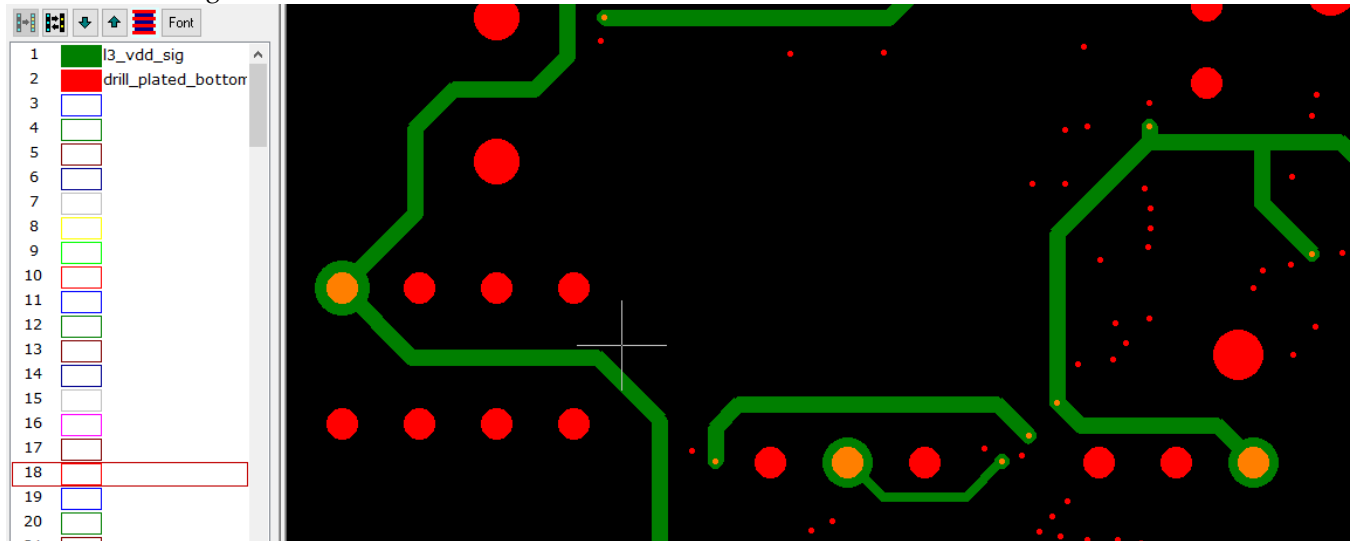
11.22.48

- Render Drill and Rout Layers as Scratch

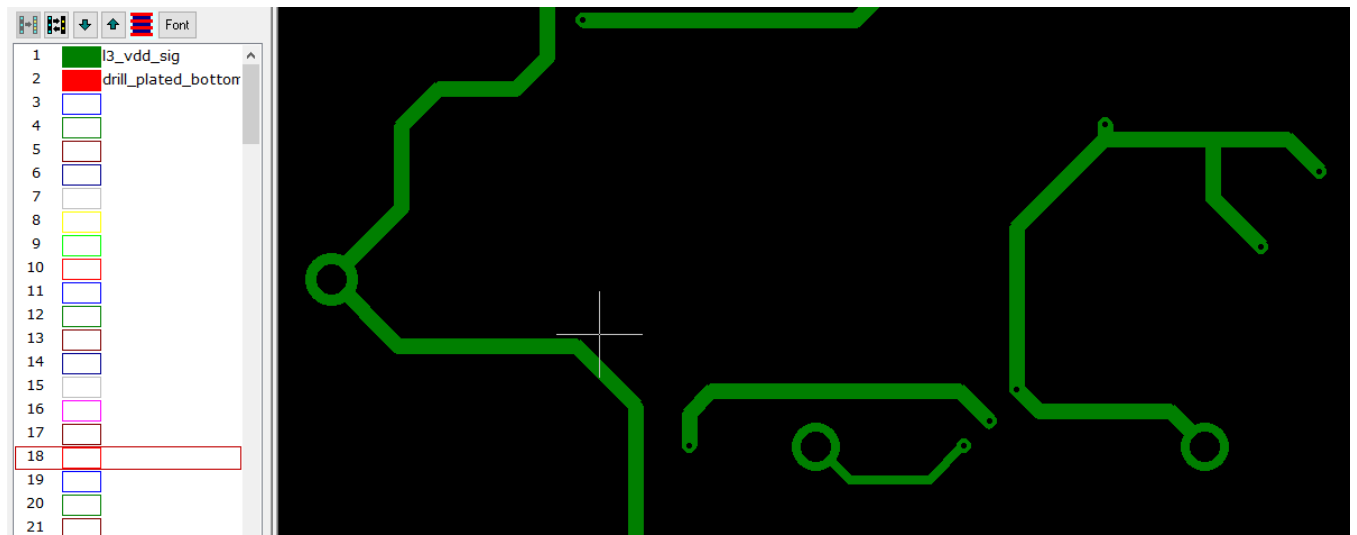
“Setup > Render Drill Layers as Scratch”.

If checked, will cause the drill and rout layers to render as scratch over all the other layers.

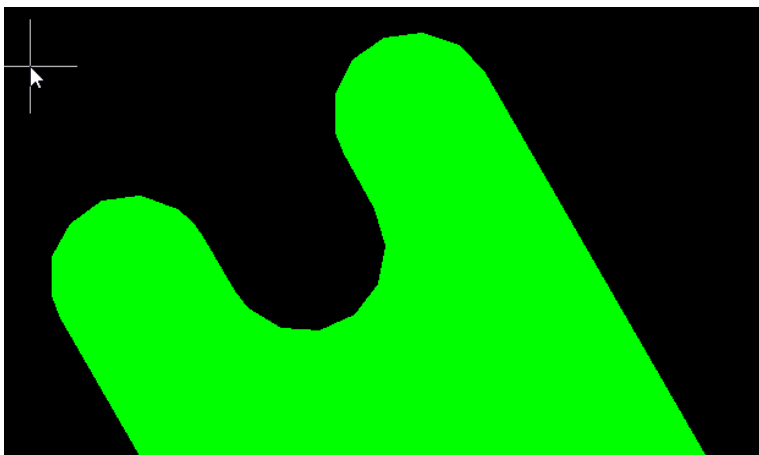
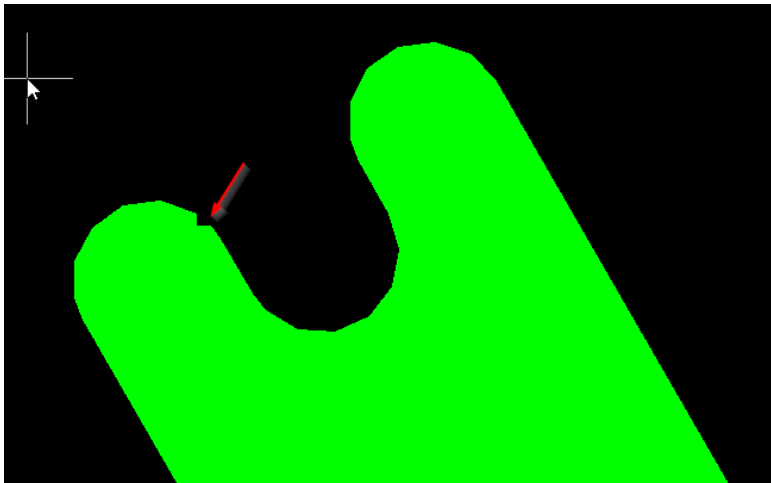
Normal rendering:



Scratch rendering:



- **New Method to Remove Dents in Polygons**
 “Edit > Edit Selection > Polygons > Remove Dents”.



- **3D Viewer Now Supports Stencil Layers As Well**
 Layers of type STL/STU..Needs new version of the 3D viewer executable as well (https://downloads.pentalogix.com/3DBoardViewer64_Setup.zip)

11.22.41

- **Layer Type in Layers Toolbar**
Display can not be controlled with “Setup > Preferences > Show Layer Type In Layers Toolbar”.
- **Polygon Chords to Arcs**
Algorithm improvement.

11.22.40

- **Select By DCode**
Now has add and remove operations, like the other selection tools.
- **Convert to Positive Operation**
Has been improved.

11.22.38

- **Select Short Traces**
“Select > Special > Short Traces”.
Active Traces shorter than the given dimension are selected.
- **Layer Type in Layers Toolbar**
The layer name is now followed by the 3 letter layer type except for the default (SIG).

11.22.37

- **New Selection Tool**
“Select > Select By Dcode Tool”.
This is a tool to select an element and all active elements of the same D Code as the one selected.
Once the tool is activated, click on an element (pad, trace) of interest.

11.22.30

- **EMS Magician Is Now Part of the Installation**
It needs no additional licensing.
“Tools > EMS Magician” or “View > Toolbars > Additional Tools”.
- **New Gerber Export Option**
It appears that a lot of other software packager and machines that consume Gerber can still not accept the latest specifications in the Gerber syntax.
To help exporting for such targets a new option has been added.

Gerber Options For Export

File Format	Data Format	Fields
RS274-X Options	Polygons	Miscellaneous

☐ Output Thermal primitive (7)
☒ Output Step And Repeat codes (SR)

☐ Output time stamp header
☐ Output LN (Layer Name)

 Negative layer is "IPNEG" followed by:
☐ LPD ☒ LPC

☐ Output rotated and complex shapes as outline

This will use the AM outline (code 4) to export complex apertures.
 This appears to be acceptable to most older software.
 This option is only relevant in "V1" export mode:

- ☐ X2 (Gerber Extended V2) (Recommended)
☒ RS-274X (Gerber Extended V1)
☐ RS-274-D (Basic Gerber - no shape info)
 (This format has been revoked by the Gerber standard)

11.22.21

- **Gerber Export of Non-standard Apertures**

They are now exported in different formats, depending on whether Gerber Extended V1 or V2 is selected in the Options dialog. If V1 option is selected only aperture macro (AM) features that existed in V1 are used.

11.22.16

- **More Efficient Export of Rounded Rectangle D Codes in Gerber**

- **Import ODB++**

Added fonts: "simplex" and "datecode". Dynamic text \$WW (week number) and \$YY (year).

11.22.12

- **Components Text Rendering for Rotated/Mirrored Components**

New option added to show text horizontally regardless of component rotation or mirroring. "Components > Show Text Horizontally".

11.22.10

- **D Codes Table – Rounding Values**

Additional rounding option added.

- **Macro > Run**


Replaced shortcut F11 with Ctrl+F11 to prevent users pressing F11 accidentally when trying to press F10.

11.22.1

- **Rotated D Codes**

D Codes can now be rotated.

The Setup D Codes dialog now shows an “Angle” that can be set.

 **D Code Table**


Edit Select Shape Dimensions... Layers... Operations Goto Custom Apertur

D Code	Shape	X	Y	Traces	(Arcs)	Pads	Angle
*D10	C	0.05	0	0	0	1	0°
*D11	S	0.05	0	0	0	1	0°
*D12	R	0.05	0.1	0	0	1	0°
D13	C	0.01	0				0°
*D14	Q	0.15	0.08	0	0	1	0°
*D15	O	0.05	0.12	0	0	1	0°
*D16	H	0.15	0.03	0	0	1	0°
*D17	T	0.2	6	0	0	1	0°
*D18	D	0.1	0	0	0	1	0°
*D19	E	0.1	0	0	0	1	0°
D20	C	0.05	0				0°
*D21	L	0.12	0.05	0	0	1	0°

You can set this angle by double clicking on the cell, or selecting any number of D Codes and doing “Operations > Rotate”.

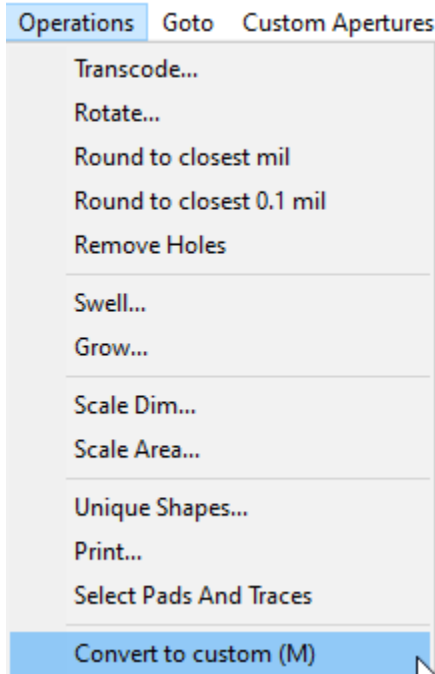
D Code	Shape	X	Y	Traces	(Arcs)	Pads	Angle
*D10	C	0.05	0	0	0	1	0°
*D11	S	0.05	0	0	0	1	0°
*D12	R	0.05	0.1	0	0	1	30°
D13	C	0.01	0				0°
*D14	Q	0.15	0.08	0	0	1	0°
*D15	O	0.05	0.12	0	0	1	0°
*D16	H	0.15	0.03	0	0	1	0°
*D17	T	0.2	6	0	0	1	0°
*D18	D	0.1	0	0	0	1	0°
*D19	E	0.1	0	0	0	1	0°
D20	C	0.05	0				0°
*D21	L	0.12	0.05	0	0	1	0°

OK



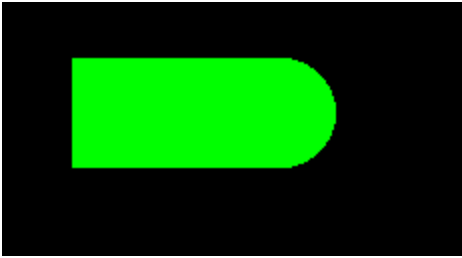
When saving a .bin file, rotated D Codes are converted to custom (M) shapes, to make the output backward compatible with previous versions of the CAM software. When opening these files in the current version, the customs will be converted to rotated built-in shapes.

One can also convert selected rotated D Codes to custom (M).



- **New built-in D Code Shape**

“Bullet” (single character designator “L”). Also known as hal-obround. Same as the obround shape (O), but with only one rounded end (to the right). Other orientations can be achieved using the “Angle” feature (see above).



- **Select by Range of Values**

“Select > Special > Pads in Size Range” and “Select > Special > Area Range”
Selects elements with size values in a range of two values.

11.20.29

- **Insert Rotated Rectangle**

“Insert > Rectangle”.

If Shift is pressed when picking the first corner, the user will be asked to click on 2 more corners. These will define a possibly rotated rectangle.

- **Consolidate Scratch Holes**

“Edit > Paint And Scratch > Scratch Holes To Polygons”.

In many cases, when data comes from Gerber files, holes in plane surfaces are represented as scratch polygons. This results in groups of paint and scratch data,

This new command tries to consolidate the holes as part of the polygons thus eliminating scratch elements.

11.20.25

- **DFM Hole Spacing**

The drill violations check now has rules for checking edge to edge spacing between drill holes.

- **DFM > Save XML Report**

Save an XML file with the violations found in the last operation.

11.20.19

- **Setup > Save Parameters to File**

Saves the parameter values of some of the commands. These can be restored later from the saved file. A preliminary set of parameters are now saved. More will be added later.

- **View > Selection > D Codes**

In visible layers only.

- **Select Drill Holes and Routing Paths**

“Select > Drill And Rout”. Use this to select elements of chosen tool numbers in a set of layers.

The command works on visible layers only. So, to restrict selection to certain layers turn off the others.

Tools numbers can be picked from the dialog that is displayed.

- **Option to Control Info to Clipboard**

“Setup > Preferences > Info To Clipboard When Selecting”.

- **Last Used Option for Transcode D Code Dialog**
Allows quick usage for the last D Code used for transcoding.
- **Added Centroid Info**
“Components > Save Centroid File” also saves the width and height of the component (non-rotated).

11.20.16

- **Extra Info When Selecting a Pad or Polygon**
When selecting just one pad or one polygon, the following info is saved in the Windows clipboard: width, height, perimeter, area.
This can then be pasted into a text document.
- **Import ODB++**
Better handling of complex jobs.
- **Connector Arcs for Routing Paths**
The setting was global for all routing paths. Now changed to be specific for individual routing paths. Applies to compensated paths only.

11.20.10

- **New Layer Visibility Action**
For the layers toolbar. The new action is Ctrl+Shift+Double-click. This will make visible ONLY the range of layers between the previous active layer and the layer double-clicked.
This is in addition to the previously available actions:
 - Double-click: toggle visibility of current layer
 - Shift+Double-click: make visible range
 - Ctrl+Double-click: turn range visibility off
- **Optional Tolerance Text**
In “Insert > Loc and Dim Setup”. If tolerance text is entered it will be added to the dimension text.
- **Concentric Pocket Routing**
An option has been added to “Tools > Routing > Pocket Routing to allow concentric routing with a single path, starting from the center of the shape and going outward.
- **Selecting SMD Components**
“Components > Select SMD Components” will select components from CMU or CML layers that appear to be SMD components, by checking against corresponding paste, mask and drill layers.
If a paste layer is present, components with matching paste pads are considered SMD.
If a paster layer is not present, the mask layer will be considered. Components will be considered SMD if paste pads correspond but no plated through holes (holes of type VIA will be ignored).

11.20.7

- **Boolean Operations on Layers**
“Edit > Layer Boolean Ops”, This will execute a Boolean operation (OR, AND or MINUS) on two layers.
The result will replace the first layer. The resulting elements will be polygons.

- **Insert Barcode**

“Insert > Text” now has a number of barcode “fonts”. Their name includes (Barcode).

The only relevant items for inserting a barcode are “Height” and “Aspect Ratio”. For barcode types that are square (like QRCode), “Aspect Ratio” will be ignored and “Height” will be the size of the generated barcode. For the others height will be given by “Height” and width by “Aspect Ratio” X “Height”.

- **New Clipping Operation**

“Edit > Break Using Line”. Similar to “Break Traces and Clip Polygons”, but uses a line segment to break instead of a frame. For polygons, the line segment needs to completely cross the polygon (cannot start from inside the polygon).

11.20.4

- **Layer Organizer**

Now supports up to 24 layers,

- **Select Text Based on Content**

“Select > Special > Text”. This will select all text blocks that match the regular expression you enter.

It uses regex, which is a bit more complicated than regular text, but allows for flexible searches.

You will need to learn just a few patterns. For instance:

- ABC will match all the text blocks that are exactly ABC

- ABC.* will match all text blocks that start with ABC. The dot character in regex matches any character and star means one or more.

- .*ABC.* will match all text blocks that contain ABC anywhere.

Be careful when you want to match an actual dot or star or any other character that has meaning to regex.

In cases like these you need to escape the character with a backslash.

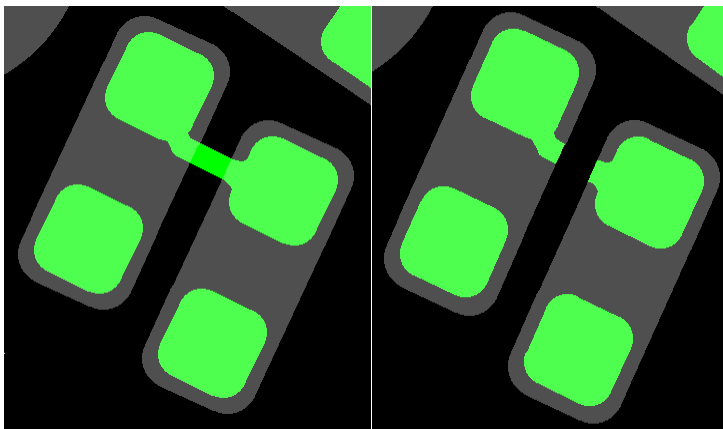
For instance ABC\. will match all blocks that are exactly “ABC.”.

There are many regex tutorial on the web,

- **Clip One Layer by Another**

“Edit > Region Clipping”. Elements in the first layer will be clipped by elements in the second layer.

The result in the first layer will be the clipped data as polygons. Elements that do not intersect the second layer will be deleted.



- **Tombstone DFM**

“DFM > Tombstone Violations”. The concept of tombstone violations is described here:

<https://www.eurocircuits.com/pcb-assembly-guidelines-tombstoning/>.

Note that these will only be found if the job has component layers (CMU/CML).

These are usually imported from ODB++, IPC-2581 or Gerber X3. One can also generate them manually in

the CAM software.

When the silkscreen option is also selected, components that have silkscreen under them will also be flagged as violations.

- **Time Added to Rout Length Statistics**

“Tools > Routing > Length Statistics”. In addition to the length, the time to rout the length is also shown. This applies only to tools that have the “Table Feed” parameter non-zero in the Tools Setup dialog.

- **Routing Plunge Point**

New command “Tools > Routing > Set Plunge At Mid Edge”. Adds the plunge point at the middle of the edge (segment) close to the click.

11.20.1

- **Import IPC-2581**

Files in IPC-2581 format can now be imported.

- **Reset All Routing Path Precedences**

“Tools > Routing > Precedences”. Sets all precedences for the selected routing paths to “don’t care”. Useful before sorting the routing paths for export if existing precedence should be disregarded.

11.18.52

- **Zoom on Selection**

“View > Zoom > On Selection” or “Ctrl + Shift + Home”.

- **Double Hits DFM**

Drill DFM now has a double hits check for finding duplicate drills with the same tool at the same position.

11.18.51

- **Support for Gerber Job Files (.gbrjob)**

Gerber job files contain info a set of Gerber files that constitute a job. The main info is the mapping of file names to layer numbers.

If a .gbrjob file is present, one should import (or open) that file. The corresponding Gerber files will then be loaded.

Gerber job files can also be exported. For this you need to pick the “Layer by layer” mode and X2 as the Gerber dialect (in Options).

11.18.49

- **Enforce Min Spacing for Mask/Paste Pads**

“Tools > Soldermask/paste > Enforce Min Spacing”. Select a group of pads for a component and specify the min desired spacing between pads. If the spacing is less, it will be corrected with scratch elements.

This can be applied to pads on any layer, but is most useful for paste layers.

11.18.46

- **New Component Operations**

“Component > Select”. Can select/deselect CAM components and pads with component info.

“Component > Add Component Info to Pads” can be used to add component info from component layers (CMU and CML) to the corresponding layers with pads: copper, soldermask and solderpaste (CPU/CPL, SMU/SML, SPU/SPL). Only visible layers will be processed.

- **Support for Gerber X2 Component Info**

Support for “TO.C” (reference designator) and “TO.P” (pin name)”.

- **Shortcut for “Go > Center of Selection”**

Ctrl+Shift+K.

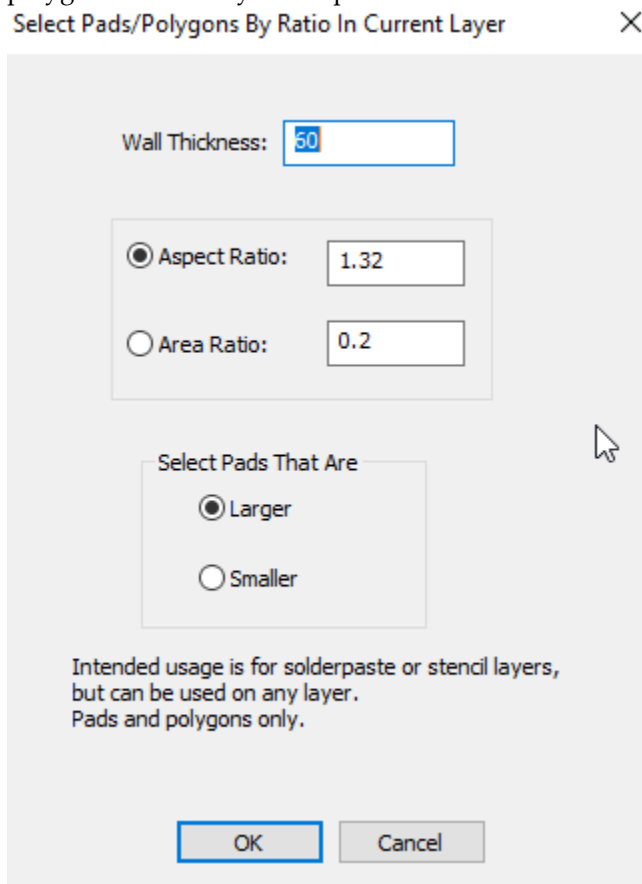
- **“Go > Coordinates” improvement**

One can now type arithmetic operation in the X and Y boxes. For instance: “10.2+0.25” or “230.1 / 1.25”.

Only two terms allowed and no units in this case.

- **Ratio Validation**

“Tools > Validate Ratios”. Useful for stencil and paste layers. Works on current layer by selecting pads or polygons that satisfy the requested rule.



Computation of ratio is based on thickness and the bounding box of each element.

If dim is the dimension of the bounding box the formulas are:

Aspect ratio = (smallest of dim.x and dim.y) / thickness

Area ratio = (dim.x * dim.y) / (2 * (dim.x+dim.y) * thickness)

11.18.44

- **Converting Pad Patterns to Components**

The “Components” menu has new items: “Convert Footprints to Components”, “Change Reference Designator Tool” and “Change Orientation Tool”.

To see how to use this feature read the document “Convert Pads to Components.pdf” included in the “Documents” folder of the installation.

11.18.40

- **Changing Date in Text**

“Edit > Edit Selection > Text > Change Date”.

This applies to selected text blocks that contain a date. The new operation allows changing the date to the current date (today).

The date needs to contain year, month and day, but the order is arbitrary.

The user is prompted with the format: “YYYY-MM-DD”. The string can be changed if the order in the text does not match. For instance one could input “DD-MM-YYYY”.

Note that the separator (“-”, in this case) does not need to be the same in the text, but the new replacement date will use it.

For instance: “YYYY-MM-DD” will match “2021-03-02” or “2021 03 02” or “2021:03:02”. Any separator as long as it is the same. However, the replacement date will use “-”.

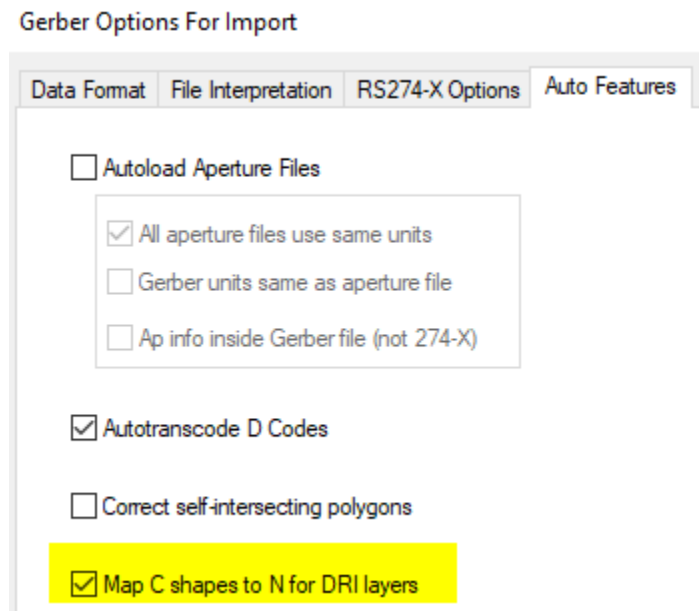
- **Insert > Chamfer**

Reverted to previous behavior where traces that are not really part of the chamfer can be moved.

- **Import Gerber Autoconvert C to N**

This was previously the default for DRI layers or when importing files with the “.dri” extension.

Not this feature can be turned on or off:



11.18.37

- **New Delete Vertex Operation**

“Edit > Edit Traces > Delete Vertex”. A tool to delete the vertex common to two straight traces. Clicking

on the vertex deletes the two traces and replaces them with a spanning trace (endpoints equal to the other endpoints of the removed traces).

11.18.36

- **Improved Display of Selected D Codes and Tool Codes**

“View > Selection > D Codes” and “View > Selection > NC Tools” now includes a count of elements also.

- **Edit Custom Font**

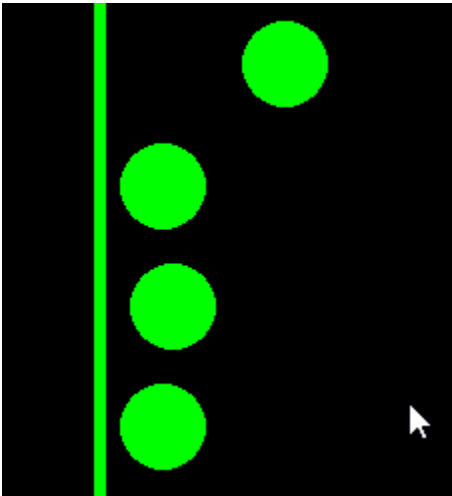
Read the document “Font Editing.pdf” in the installation folder.

11.18.31

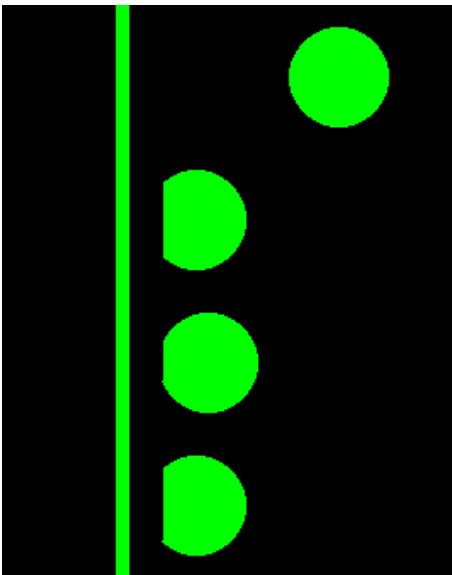
- **New Shave Pads Command**

“Edit > Shave Pads > Shave Pads vs Trace”. Select a trace and any number of pads next to it. This new command will shave the pads to insure the desired gap.

Before:



After:



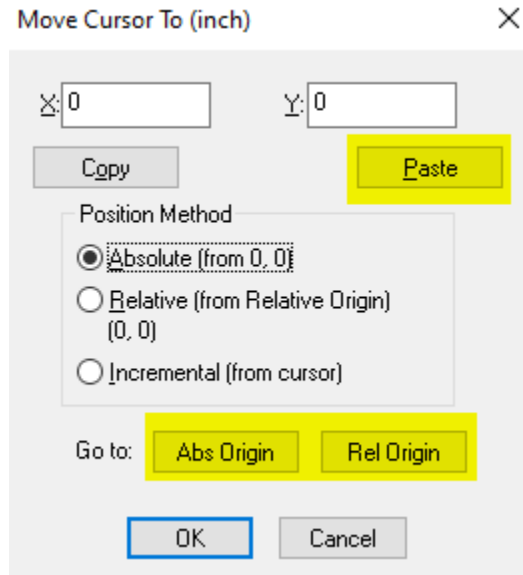
11.18.28

- **“Select > Connected”**
Added “add” (Shift key) and “remove” (Ctrl key) modes.

11.18.26

- **New Functionality in “Go > Coordinates”**

Several new buttons have been added.



The “Paste” button will paste the text currently in the Windows clipboard into the X/Y fields.

This is useful mostly for copying a pair of coordinates, for instance from an Excel spreadsheet. The pair should be separated by spaces. The two items will be pasted into the X and Y fields.

The “Origin” buttons will move the cursor to the absolute origin (0,0) or relative origin (if set).

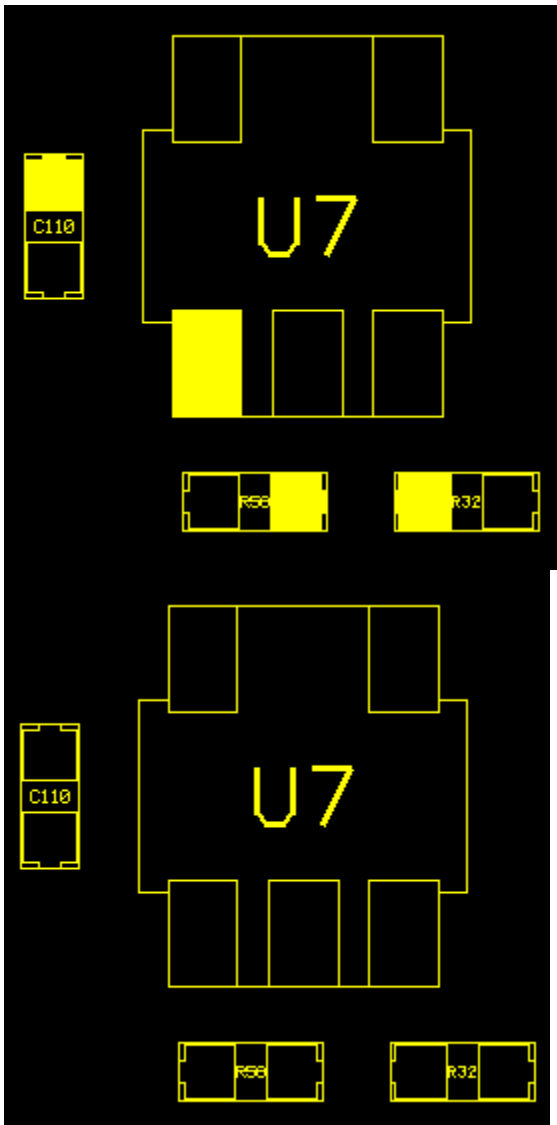
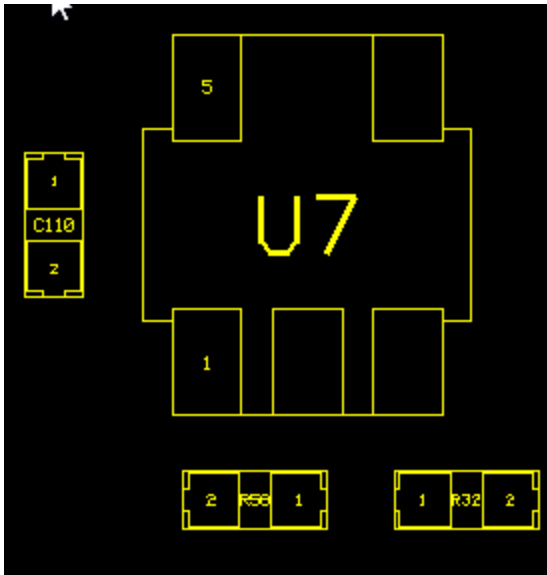
- **Improved Display of “Elements Overlapping Crosshair”**

11.18.20

- **Mirror X Option for Print**
Useful for printing images for the bottom side of a board.

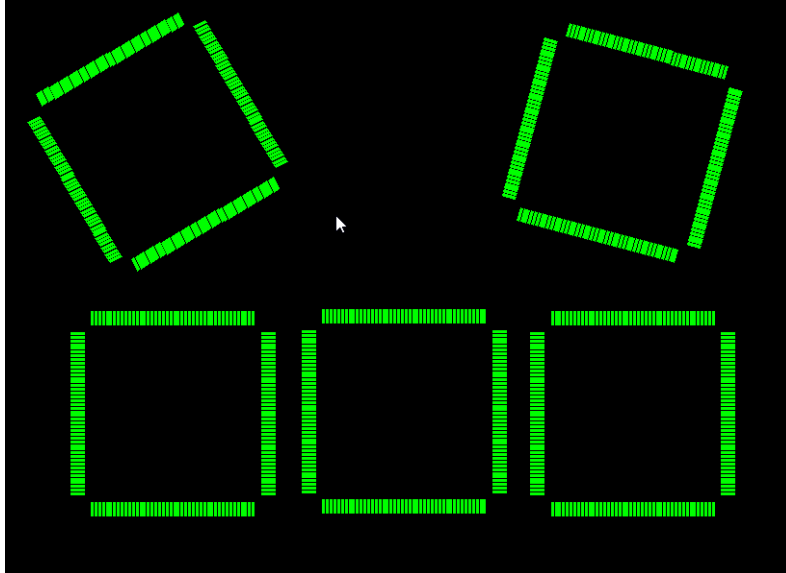
11.18.19

- **Elements Overlapping Cursor**
The dialog showing info on the elements that overlap the cursor now show more detailed information.
- **Import GDS-TXT**
Now allows for several files to be imported in one operation.
- **Pin 1 Display Options for Components**
“Components > Pin 1 Rendering”. Has 3 options: Text, Filled, None. Previous way of rendering was None.
Examples are shown below for the 3 modes:

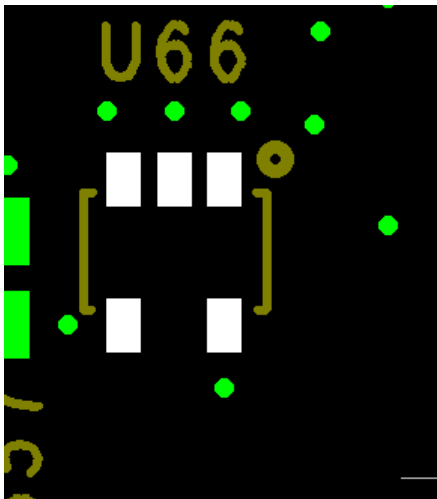


11.18.12

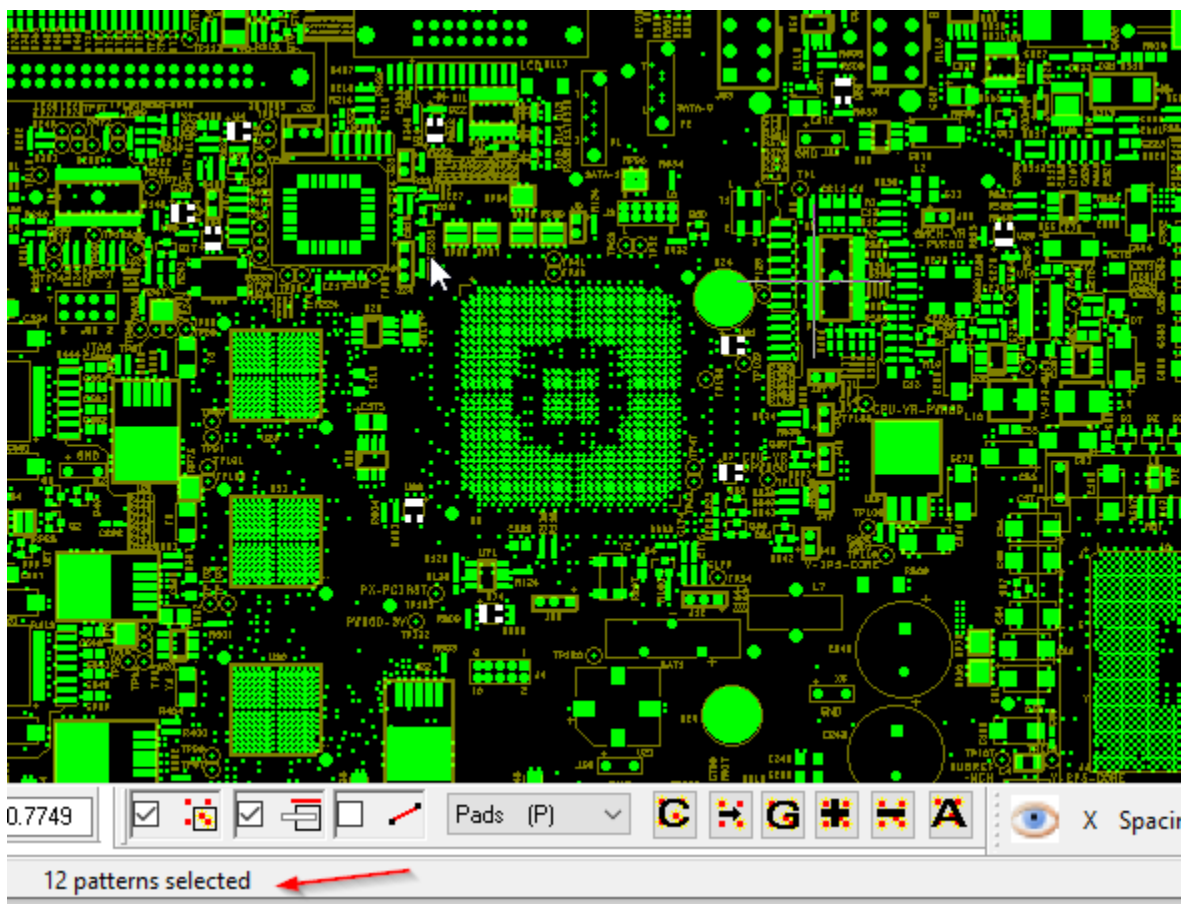
- **Select Invalid Polygons**
“Select > Special > Invalid Polygons”. Selects polygons that are not valid according to the Gerber or ODB standard (are self-intersecting).
- **Replicate To More Than One Layer**
“Edit > Edit Selection > Replicate” now allows selecting more than one layer to replicate to. This is possible only if the selection to replicate is in one layer only.
- **Stencils Can Be Generated for Rotated Components As Well**



- **Netlist Computation Improvements**
Phase nets computation (for Emma testers) has been improved.
- **Footprint Polygons from Component Data**
“Components > Generate Footprint Layers”: if component layers are present (CMU, CML), usually from ODB++ data, one can generate polygons for the footprints (pins) in new layers.
- **New Method to Find Component Footprints.**
“Components > Find Similar Footprints”.
One first needs to select the pads belonging to a component. Then use the above command. Similar arrangements of pads in the same layer will be selected.
Note that this works for any pattern of pads, they don't need to belong to a component.
An example: Select the pads as shown below from the soldermask layer:



After the executing the command, the screen will show:



12 additional such patterns have been found.

The silkscreen layer need not be present. Only shown for illustrative purposes.

11.18.10

- **3D Board Viewer**

Added menu item to select layers to display.

- **HomeBase Stencil**
Can now process rotated rectangle pairs.

11.18.7

- **Select Off Center Pads**
“Select > Special > “Off Center Pads in Two Layers”. Selects pads in two chosen layers. A pad will be selected if it overlaps a pad in the other layer, but has a different center.

11.18.3

- **3D Viewer Renders Paste Layers Also**
Layers of types SPU/SPL are also rendered. In addition, only visible layers are rendered, so one can choose what is rendered in 3D.
A new version of the 3D viewer is also needed.
Download from here:
http://downloads.pentalogix.com/3DBoardViewer64_Setup.zip
- **Pocket Routing Generates One Path Only**
“Tools > Routing > Pocket Routing” will now generate only one continuous path.
- **Select Pads with Plated Holes**
“Select > Special > Pads With Plated Holes” will select pads in a copper layer that have corresponding plated drill holes in the drill layers in the current job.

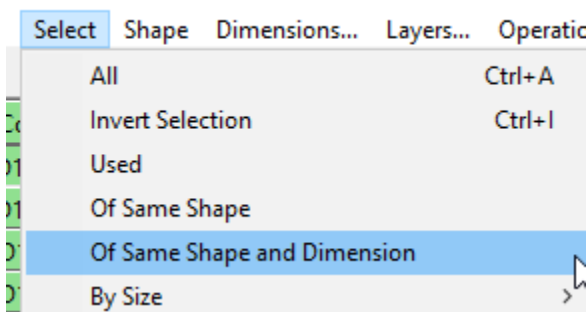
11.18.1

- **Import and Export Gerber Components**
Support for the new Gerber components format (called Gerber X3) has been added.
To export component layers (of type CMU and CML) one needs to pick “X2” as the format from the Options dialog.

11.16.73

- **Changing Shapes of D Codes in Different Orientation**
There is a new command in the D Codes Setup dialog that selects all D Codes of the same shape independent of orientation.

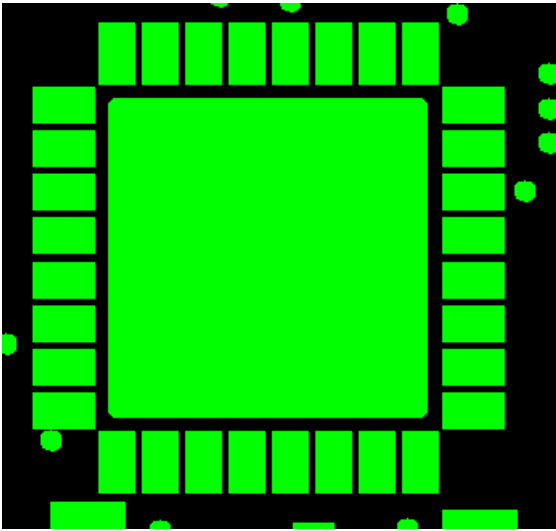
D Code Table



This can be used to select all D Codes that have the same shape type and dimensions, independent of orientation.

One common use case is the following.

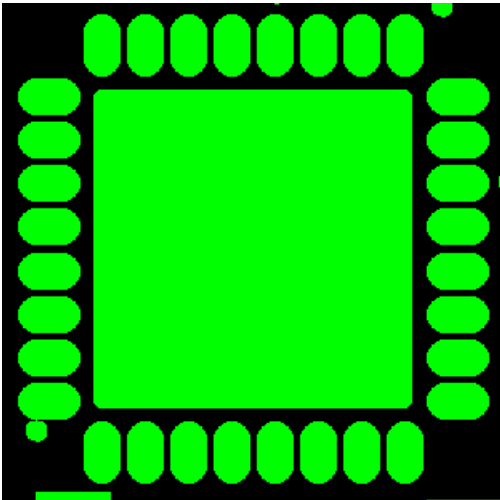
Assume you have pads as shown below:



and you want to convert the rectangles to obrounds.

In the Setup dialog you can use the above command to select the two D Codes (or even select them manually. Then press “O” for Obround. When the dimensions dialog comes up click OK, although you could also change the dimensions if you wanted.

The result will be:

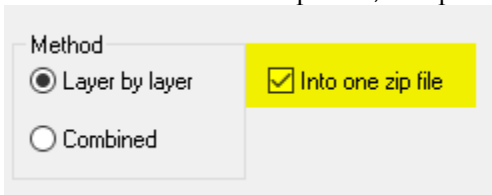


- **3D Viewer Supports Blind Vias**

Previous version ignored blind vias and they did not appear in the 3D view. A new version of the 3D Viewer executable is also needed (V3.0 or higher).

- **Export to Zip File**

When several files are exported, an option has been added to export them as a zipped file.



- **Export ODB++**

New options for exporting N shape pads (holes). The options for ODB symbol are “hole” and “round”. “round” should be picked when exporting for software that canoe process “hole”.

11.16.66

- **Several Files In Open Dialog**

One can now pick more than one file in the “File > Open” dialog. Of course, this is mostly useful for import, not for open. One should open only one .bin file.

- **Improved Phase Net Data**

For Contour netlist extract. The generation of phase test data has been improved.

11.16.63

- **Larger D Code Numbers Allowed**

Largest legal D Code number has been changed to 99,999. Previous value was 9,999.

11.16.61

- **Expand or Contract a Closed Loop of Traces**

“Edit > Edit Selection > Special > Expand Closed Trace Contour”

The selection should contain a closed loop of traces of the same D Code.

This command will expand or contract this loop, similar to a polygon swell operation, but without the need to convert to a polygon and back.

Useful for changing board boundaries and other boundaries.

11.16.53

- **File Import Dialogs are Now Resizable**

- **Import DXF Splines**

Splines are now approximated with straight segments. However, the rendering may not match exactly the one seen in AutoCad in all cases.

- **Custom DFM Soldermask Checks**

More checks have been added: Minimum Web, Inner Pad to Mask and Outer Copper to Mask. To disable any of the checks, set the value to 0.

11.16.46

- **Layer Visibility**

The Layers Toolbar right-click menu has a new entry “Stays Visible” which can be checked for a layer so that it remains visible while the “Next” and “Previous” buttons are pressed.

Useful when one wants one or more reference layers to be always visible while navigating through the rest.

- **Routing Path Lengths**

“Tools/Routing/Length Statistics” shows a per tool path length for each tool in visible layers.

- **Pocket Routing Improvement**

“Tools/Routing/Pocket Routing” now handles polygons with holes as well.

- **Vertical Adjacency**

The vertical adjacency determination on netlist extract has been improved.

11.16.43

- **Repeating Last Move**

New menu item “Edit/Edit Selection/Repeat Last Move”.

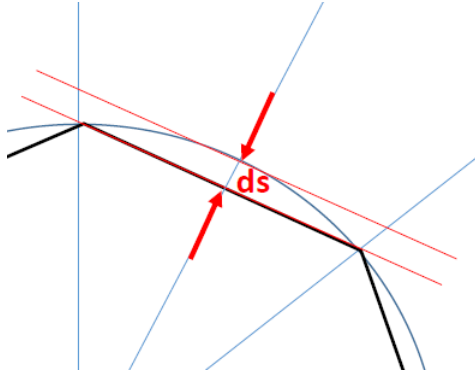
This will be active once a move operation was done. After that, this command will do a move on whatever is selected using the displacement of the last move.

Useful when one wants to move several separate elements by the same amount.

- **New Arc to Chords Parameter**

For all the operations that convert arcs to chords a new parameter has been added, sagitta.

It is the distance shown as “ds” in the diagram below:



This is a good measure as to how well the chords approximate the arc.

- **Insert Text Aspect Ratio**

The insert text dialog has a new field called “Aspect ratio”. A value of 1.0 will have the nominal text scaling, Values larger than 1 will compress horizontally, while smaller than 1 will expand.

- **Export ODB++**

Now exports only visible layers. Previously exported all loaded layers.

- **Moving Center of Custom Shapes (M)**

Sometimes one needs to move the center of M shapes that have irregular shapes.

To do this, select one pad with the D Code to change, then move the cursor to the location for the desired center. Then do “Edit/Set M Shape Center to Cursor”

11.16.38

- **Context Menu now shows last two commands**

Right click mouse menu shows on top the two last used commands. Previously only the last one was showed.

11.16.34

- **Rotate and Mirror Transcode Report**

A new option in the dialogs has been added to generate a text file report of the D Codes that were trans-coded.

- **Job Report**

“File/Report/Generate” now also includes a per layer D Code report.

11.16.26

- **Offset Polygons**

“Edit/Edit Selection/Polygons/Offset”.

New operation to inflate or deflate polygons. Similar to swell, but will avoid generating new arcs. For the swell operation the distance from the new contour to the old one is always the specified swell amount.

For offset this is not case at the vertices.

11.16.22

- **Select Elements of a Given Tool Number**

“Select / All NC T#”.

Will prompt for a tool number and then select all elements (pad, trace, routing path) mapped to that tool number. The shortcut key for this is Ctrl+Shift+T.

- **New Options for Netlist Extract**

- “Intersecting soldermask allowed”. If checked soldermask openings that intersect a pad (or hole) will be considered valid and a default soldermask opening will be assumed. Otherwise the pad or hole will be considered covered by soldermask.

- “Missing Soldermask Layer”. Controls how soldermask is interpreted if a soldermask layer is missing (no MSU or MSL layer present). One can pick one of two choices: the corresponding components layer is completely open (no soldermask) or completely covered (soldermask everywhere),

11.16.16

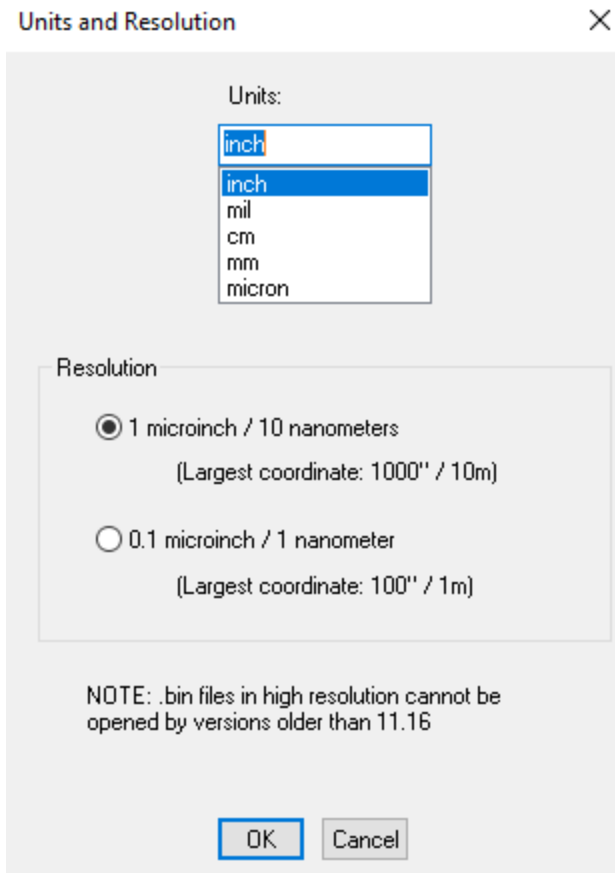
- **New Units and Resolution**

The units selection now allows for “micron” as well as the other previous units.

When specifying the units with text, one can use “micron” or “um”.

There is now also a way to specify the internal database resolution. This is independent of the units.

To do this use “Setup / Units”:



Note that for PCB applications, the "1 microinch" resolution is more than adequate. This was the resolution of previous versions of the software. The new "1 nanometer" resolution is needed when processing chip CAD data.

11.16.12

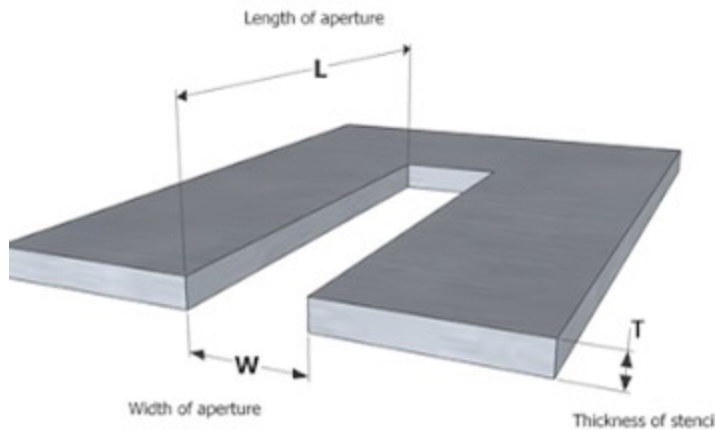
- **Stencil Ratios Computation**

"Tools/Stencil Apertures/Ratio".

These ratios are computed on the selected elements. Need to select a stencil (or part of a stencil first).

Also, one needs to specify the thickness of the "wall" before getting the ratio (for "Area Ratio" and "Surface Area Ratio").

The formulas for "Aspect Ratio" and "Area Ratio" are below:



$$\text{Aspect Ratio} = \frac{\text{Aperture Width (W)}}{\text{Stencil Thickness (T)}} > 1.5$$

$$\text{Area Ratio} = \frac{\text{Area of Pad (L x W)}}{\text{Area of Aperture Walls (2 x (L+W) x T)}} > 0.66$$

For “Surface Area Ratio” the formula is based on the exact shape of the aperture: area of aperture walls divided by the area of the aperture.

- **Polygon Cleanup**

A new command has been added: “Edit / Edit Selection / Polygons / Combine Short Straights”. Given a tolerance value, any pair of successive straight edges where each is shorter than the tolerance will be replaced with an edge connecting the outside vertices of the pair.

- **Export DXF**

Files with step and repeat will be exported using BLOCK and INSERT, resulting in smaller output files. Only top-level step and repeat blocks will be exported this way.

- **Import/Export GDS-TXT**

GDS Binary files are currently not supported.

Only the flat part of the GDS data is imported (BOUNDARY, PATH, BOX at top-level).

Data is converted to polygons before export.

11.16.7

- **Select by Line**

A new selection tool has been added, “Select/Select By Line Tool”.

This is a tool similar to the “Select/Selection Tool” but will select elements intersecting a dragged line rather than a frame.

The usual filters are considered (Only current layer etc.).

Keyboard modifiers are: “Shift” to add to an existing selection and “Ctrl” to remove from an existing selection.

- **Copy/Paste Coordinates String**

One can now copy the text of the current coordinates to the Windows clipboard. This text can be later copied elsewhere, as needed.

There are two ways of doing this.

- By clicking on any of the coordinate display boxes in the Position toolbar (normally in the lower left corner of the application).

- By pressing the new “Copy” button in the “Go/Coordinates” box. In this case the text for both coordinates is copied.

11.16.1

- **New Insert Mode: Polygons**

The “Insert” menu has a new command “Polygons”. This will allow a user to click on a number of consecutive edges of a polygon and, when done, insert a polygon element.

To stop insertion and generate the polygon, double-click or press Enter twice.

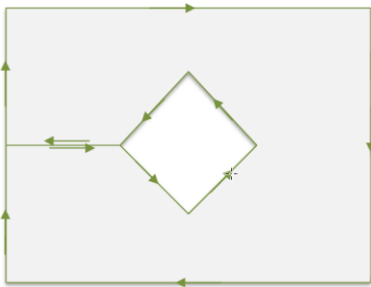
If the point where the insertion was stopped is not the start point, a connecting edge will be added back to the start point.

Previously, the only way to insert a polygon was via a several step operation:

- first one needed to insert zero width traces (“F” type D Code)
- then select them
- and finally run “Edit/Edit Selection/Polygons/F to Polygons”.

The limitation of this new mode is that one can only generate polygons with straight edges (no arcs). If arcs are needed the “F” method shown above should be used.

Note that it is possible to generate a polygon with holes also, but one need to use overlapping edges to travel to the hole and back. These are called cut-in contours:



To facilitate matching vertices in the pending contour a new shortcut has been added: Ctrl+Shift+J (jump to closest vertex of current contour).

- **Round Chamfers for Polygons**

New command to add round chamfers at polygon vertices where two straights meet: “Edit/Edit Selection/Polygons/Add Round Chamfers”.

This will replace a vertex where two straights meet with an arc tangent to both straight edges.

There are two methods for doing this:

- one can specify the radius of the arc (same radius will be used at all vertices).
- one can specify the offset from the vertex to the added arc (different radiuses will result depending on the angle of the edges)

- **Back Drill Layer Types**

Two new layer types have been added BDU (Back Drill Upper) and BDL (Back Drill Lower).

These types are only relevant for netlist extraction.

They should contain back drill pads (starting from the upper or lower component layer).

Back drilling is also called controlled depth drilling. This is a technique to remove the unused portion, or stub, of copper barrel from a through hole because when a high-speed signal travels between PCB layers through a copper barrel it can be distorted.

When netlist is extracted pads on the corresponding copper layer which contain the back drill holes will be made midpoints so that they will not be tested.

11.14.59

- **Swell By Percentage**

To swell by percentage type in a number followed by a percent sign in the “Edit/Edit Selection/Swell” dialog. The amount to swell by will be:

- the percentage of the width of symmetrical D Codes
- the percentage of the largest inscribed circle diameter for asymmetrical D Code shapes and polygons.

11.14.55

- **Stencil Sanity Check**

A user settable parameter has been added – the allowed center offset, as a percentage of the pad dimensions. The smaller the number the more errors will be reported.

11.14.49

- **New version of Layer Organizer**

This version of the Layer Organizer allows you to quickly set the color of the organized layers. You can also quickly control layer visibility to show only the top, bottom or inner layer view of your design. You can use the new “Layer Color Manager” to setup “Layer Color Templates” that make it easy for you to map each layer to one of the 15 available colors in our CAM editors.

More information on this is in the “Layer Organizer Release Notes.pdf” file in the “Documents” folder.

- **Components**

Added ability to create stencil outlines for packages and generate stencil layers from component layers.

- **Export ODB++**

Added panel size option. It is exported in the “attrlist” file as “panel_size”.

11.14.47

- **Import XNC**

Support for the XNC metadata (layer and tool info).

11.14.45

- **Export XNC**

This is an NC data format that includes drill and rout data. It is defined as “The PCB CAD to CAM Exchange NC Format”.

The format is described here:

https://www.ucamco.com/the_xnc_file_format_specification.pdf

It is based on the Excellon format, but is completely unambiguous.

No format needs to be provided by the user.

This export is a bit different from the other PentaLogix exports in that the software picks the file name(s) based on the current job name. All visible layers with NC data will be exported.

Note that this format is designed for CAD data and as such, does not support step and repeat commands.

It is best to use this to export one-up data.

However, if the CAM data does contain step and repeats the export will still work, but the data will contain multiple copies in the stepped positions.

These files can be imported by any software that accepts Excellon data, as it is a subset of Excellon.

They contain meta information about the layer type and tool types, but these are embedded in comment lines, so they will be ignored by readers that do not support XNC.

The next release of our software will also interpret this info (on import).

- **Update Available Prompt**

Users who do not want to be prompted when a new version of the software is available can now turn on the periodic check with “Help/Update Reminder”.

11.14.43

- **Setting Folder for Persistence/Configuration**

Software parameters / user preferences are saved in a file named CAMMaster.cfg or CAMMasterDsg.cfg. These parameters are then loaded the next time the software starts.

The default folder where this file is saved is the user’s “home” folder, given by the Windows environment variables “HOMEDRIVE” and “HOMEPATH”. This folder needs to be writable.

In some cases IT personnel sets these to folders that are not writable.

To mitigate this, there are two ways to tell the software which folder to use:

1. Starting the software with arguments on the command line.
“/cfg *foldername*”, where *foldername* is the path to the folder
2. Setting the Windows environment variable “PENTALOGIX_HOME” to the path of the folder.

- **Components**

1. New command to select and show a component of a given Reference Designator (“Components/Select by Reference Designator”).
2. Save a centroid file from loaded component information (“Components/Save Centroid File”).

- **Path in Layer Names**

A new option to drop the path part in layer names (“Setup/Drop Path from Layer Names”). During certain import operations the layer names will contain the full path. When this option is checked, the path part is dropped and only the file name is kept.

11.14.35

- **Setting Layer Colors Based on Layer Type**

“Setup/Layer Colors/Set by Layer Type”.

Will set the color of loaded layers based on the layer type. There is a default mapping from layer to layer color index but it can be modified using “Setup/Layer Colors/Restore Layer Type Map from File”.

The best way to create such a file is to first save the current one, then modify the generated file with a text editor. This is important because the layer type strings in the file are the only allowed values.

The default file looks like this:

```
CopperUpper=10
CopperLower=9
PowerGround=7
Signal=2
Drill=12
Via=15
Rout=4
BoardOutline=4
SilkscreenUpper=14
SilkscreenLower=14
SoldermaskUpper=7
SoldermaskLower=7
SolderpasteUpper=7
SolderpasteLower=7
ComponentUpper=14
ComponentLower=13
```

Note that some of strings refer to a whole set of layer types. For instance “PowerGround” refers to PW1, GD2, P25 etc.

Currently these are the only strings allowed. More can be added if needed in later versions.

The number after the equals sign is the color index in the color list that appears when choosing a color in the user interface. So, for instance, 1 is dark blue in the default setup, 2 is dark green etc.

If the colors are mapped to other RGB values, then the indices refer to the new values.

Once the map is changed, it will be remembered the next time the software starts.

- **Components**

A whole set of operations has been added for dealing with component info. It is in under “Components” in the main menu. There are too many new features to describe here.

Documentation is included in the “Documentation” folder in the installation directory.

Please look for “Components in CAM Software.pdf”.

- **Raster Mode for Compare Layers**

A new mode has been added to the Compare Layers operation. In addition to “Element by element” and “Copper area” there is now a “Raster” mode. In this mode comparison is done by using the raster images of the layers.

The advantage is that this operation is a lot faster.

The disadvantages:

- only one error is reported. This makes this mode useful when comparing two layers that are supposed to be identical and one is interested only in a yes/no answer.

- it may need a lot of memory. If there isn't enough memory available, the comparison will switch to “Copper area” mode.

- **Compare Layers by Name**

A new command has been added “Tools/Compare Layers/Compare by Layer Name”.

This is useful for comparing job data before and after. Both sets of data need to be loaded. Layer pairs with the same name are compared. Only the first error is reported.

This is most useful for checking two versions of the same data that are expected to be identical as far as geometry. It will use Raster Mode, unless the memory needed is too large in which case it will switch to copper area.

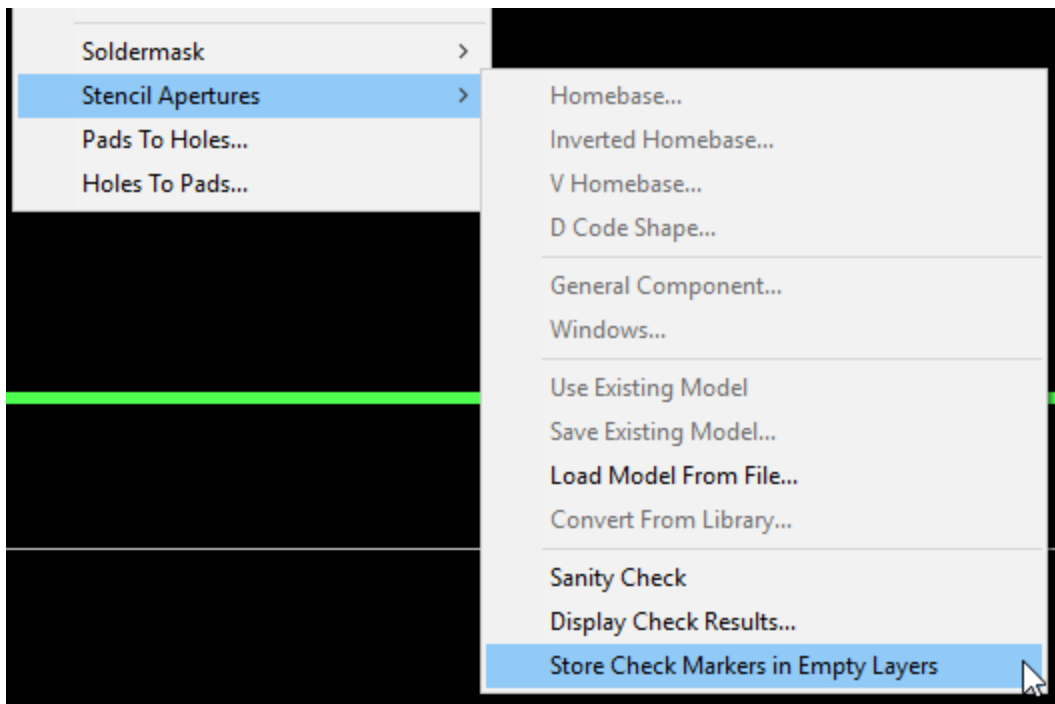
- **Import Directory Improvement**

For directories that have both Gerber files and drill files, the drill file format, if not specified, will be computed to match the Gerber files rather.

11.14.25

- **Stencil Tool Sanity Check**

The error markers can now be stored as pads in empty layers. This allows the errors to be printed if needed. They are stored as “T” shapes, but can later be remapped to any desired shape.

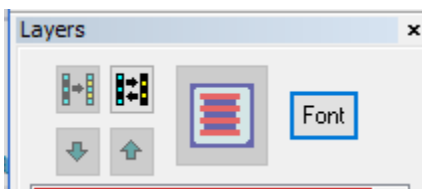


11.14.20

- **NetCompare Report**
Long list of net names is no longer truncated.

11.14.1

- **Improved ODB++ Import**
The ODB import module has been rewritten. It now supports ODB++ V8.
The import dialog has a checkbox for using the old (legacy) import module, if needed as a fallback.
- **DFM – Compute All Min Values in One Step**
New menu item “Custom DFM/Find Min All” computes minimum feature size, minimum spacing and smallest drill hole in one operation.
- **Text Size in Layers Toolbar**
On high resolution monitors, the text of the Layers Toolbar can be hard to read.
A new font selector button has been added (“Font”). This allows changing the font and the size of the font.



- **Soldermask Export for ProbeMaster**
The soldermask data exported in .f04 files has been improved to facilitate processing in ProbeMaster.
- **Reviewing Markers from a Previous Operation**
“View/Navigate Markers” will bring up again the navigation dialog from the previous operation that

generate one, like polygon corrections, layer compare etc. If you modify the data after the operation that the markers applied to, reviewing the markers will still point to the old data locations.

- **Loaded Layer Marker in Layers Toolbar**

The “*” marker for loaded layers has been removed. Users complained that it was hard to differentiate it from the “+” marker (multiple polarities). One can see which layers are loaded by the fact that their color bar is solid.

11.12.54

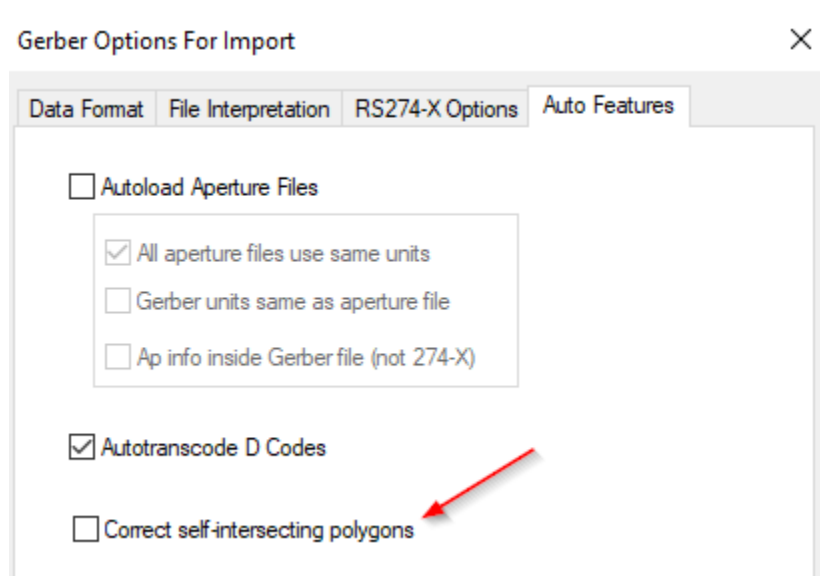
- **Gerber Import – Self-intersecting Polygons**

The handling of self-intersecting polygons has been changed. These are polygons where edges intersect at points which are not endpoints of the edges (this includes overlapping edges).

These polygons are not acceptable according to the Gerber spec.

Previously the user was prompted to auto-correct these polygons.

Now a new options has been added to the Gerber import Options dialog:



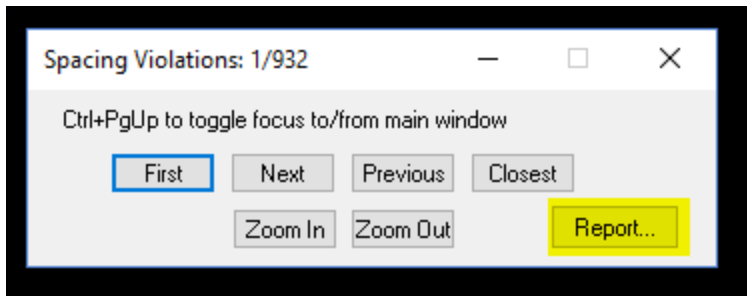
When unchecked, polygons will be left as in the import data (without prompting).

When checked, polygons will be auto-corrected and at the end of the import a dialog will pop up to navigate the places where the corrections took place (only one place per polygons is marked). This allows for a quick visual check to make sure the auto-correction behaved as expected.

Note that if the data is only loaded for viewing, or if one wants to re-export the polygons in the same state, leaving the polygons unchanged is an acceptable option.

- **Report Info**

One can now save a report after certain inspection operations, such as the Custom DFM checks, Stencil Aperture Sanity checks, Layer Compare etc. The navigation dialog has a new button called “Report”:



This will bring up a dialog with the text for all the items which can then be copied and pasted into a file, if needed.

11.12.48

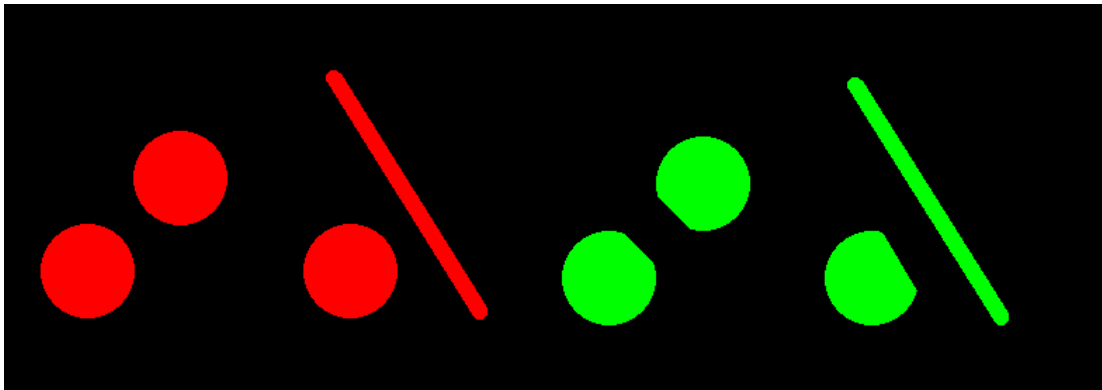
- **New Pad Shave Operation**

In addition to the “Shave Pad Tool” (in “Edit/Shave Pads/Shave Pad Tool”) a new operation has been added that shaves a pair of pads or a pad versus an adjoining trace.

This is “Edit/Shave Pads/Shave Selected”.

For this you need to select two adjoining pads or a pad and a trace and activate this command. You will be prompted for the desired gap between the two. Scratch elements will be inserted to shave the pads to create the desired gap.

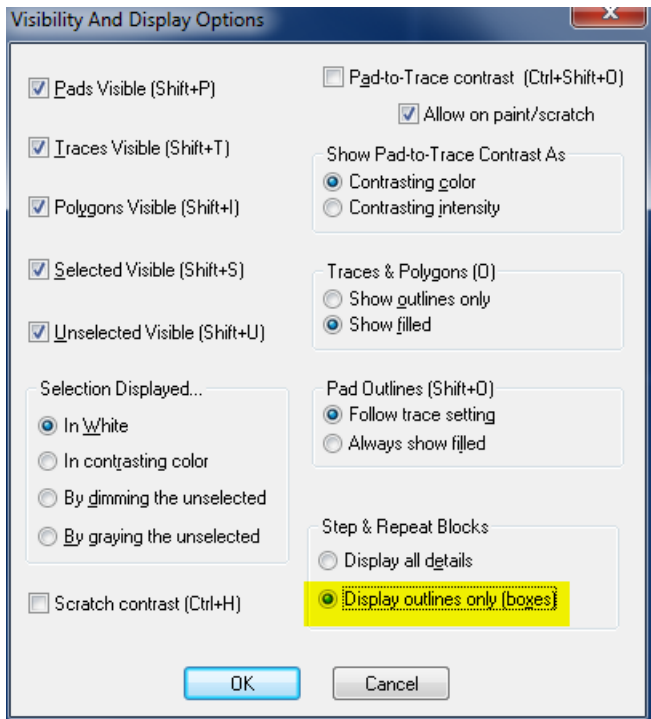
An example of before (red) and after (green) is shown below:



11.12.46

- **Step and Repeat Visualization**

When picking “Display outlines only” in the “Setup/Visibility and Display” dialog:



the behavior has changed to display elements in the first step and repeat block. Previously only a box was displayed for this block also. This gives a better idea as to what is step and repeated.

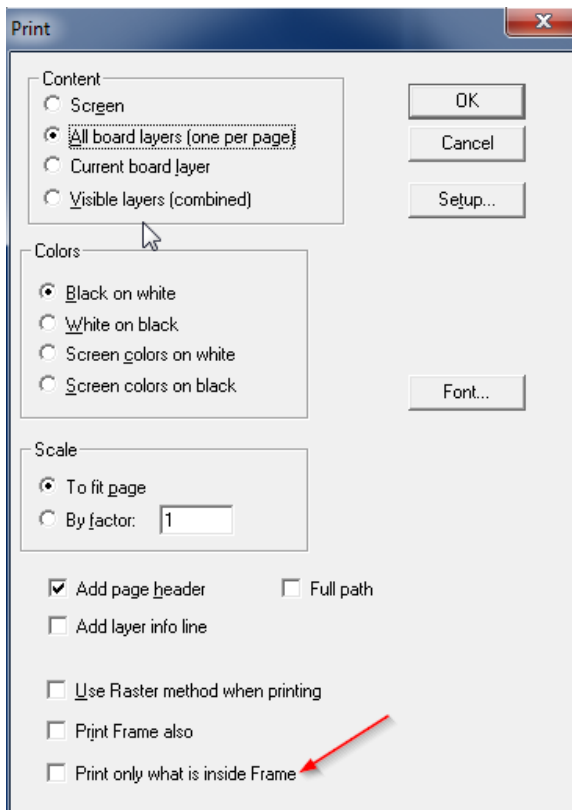
- **Drill Proximity Sort**

An new sort method has been added to start sorting from the upper-left corner.

11.12.43

- **Print Options**

A new option has been added to print only the contents of the frame:



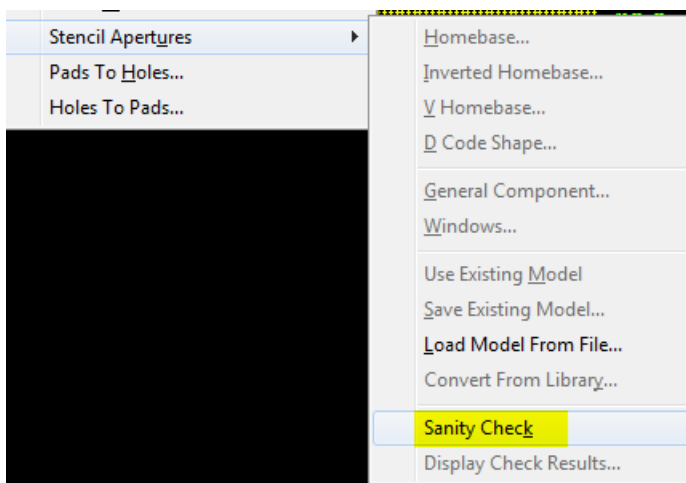
- **Gerber Export**

Export of wide arcs has been improved to account for loss of accuracy when exporting at resolutions that are lower than the internal data resolution (.6 English, .5 metric).

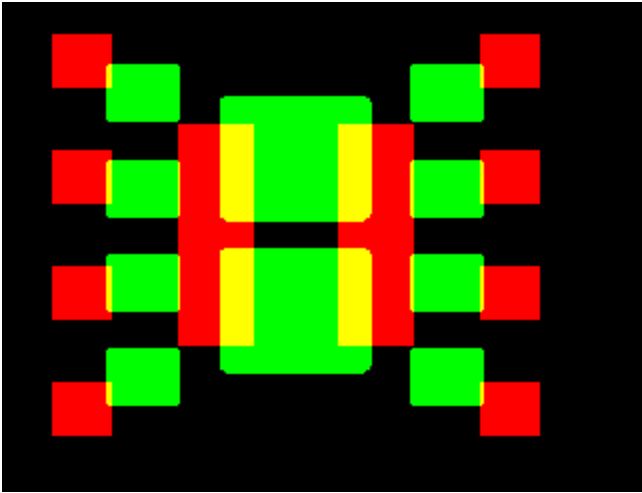
It is highly advisable to export at the highest resolution whenever possible.

- **Stencils Sanity Check**

A sanity check has been added to the stencils operations.



One first needs to make visible only two layers: the pads layer and the stencils layer. The check is done comparing the two layers. Elements are flagged which are out of place, like below:



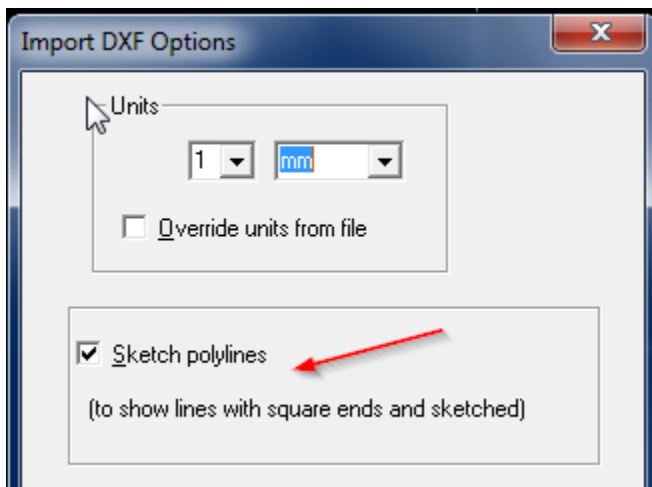
- **3D View**
Scoring layers are now visible in the 3D view (layers of type “SCG”).

11.12.40

- **Generate Images in SVG Format**
SVG stands for Scalable Vector Graphics and is a standard web format. Files with the .svg extension can be opened in Windows by double-clicking on them. They open in a web browser (Internet Explorer, etc.). The new feature is at: ‘File/Print to “.svg” File’. SVG images are useful because they can be scaled while preserving smoothness. In the web browser use the Zoom feature. In the “Layer By Layer” mode, one file is generated for each visible layer and then the collection is zipped, resulting in a .zip file. In the “Combined” mode, the image colors will look different then in the CAM software, because color blending is done differently in SVG.

11.12.36

- **New Option for DXF Import**
DXF files have lines that end with square ends. Sometimes this is the desired way of importing them. When that is the case, the “Sketch polylines” option shown below should be checked.



In most cases, when the lines correspond to circuit traces, this option should be left unchecked.

- **Gerber X2 and Netlist Info**

The new Gerber Extended V2 format (known as X2) can contain netlist info for the data elements (the net name). This is now implemented for import and export. For export you need to pick “X2” format from the “Options” dialog. For import, the netlist data is imported without any extra settings.

11.12.32

- **Import Gerber**

A warning has been added when generating data for a D Code the shape of which is undefined. Most useful for extended Gerber import, when the header does not define all the D Code shapes.

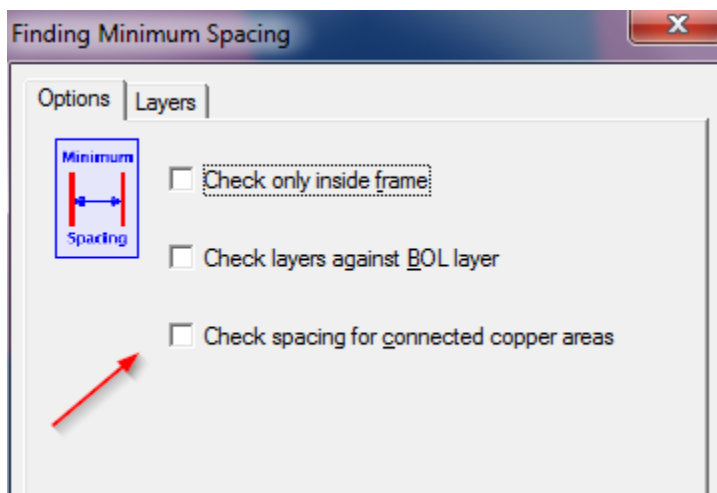
- **3D Viewer Improvement**

F type contours in rout layers are automatically converted to polygons in the drawn image.

11.12.28

- **Find Minimum Spacing**

- An option has been added to also consider connected copper areas.



- **3D View – Save Image File**

A new command has been added to the “3DView” menu: “Save Image To File”. This will save the 3D image to a file without invoking the 3D viewer. The file can later be opened in the 3D viewer, with “File/Open” or dragging and dropping.

11.12.26

- **Soldermask DFM**

“Custom DFM” now includes soldermask violations check. These checks work only on soldermask pads. If the soldermask contains polygons (for openings) they will not be considered. Checks are made for:

- Duplicate or covered soldermask pads
- Soldermask pads that are smaller than a user specified value
- Soldermask pads where the center does not match the center of the pad on the copper layer
- Soldermask pads without a matching pad on the copper layer

- **Import DXF Improvement**

If the data contains zero-width lines, the user will be prompted with an option to convert the contours to filled polygons.

- **64 bit 3D Viewer**

The 3D Viewer is now a separate install. This way users can choose the 64 bit viewer or the 32 bit viewer. The 64 bit viewer can only be used on 64 bit versions of Windows, but it can display larger boards due to not having memory limitations. Installer can be downloaded from the PentaLogix FTP site.

11.12.22

- **Improved Gerber Extended AM Parsing**

Parenthesized expressions are now allowed.

- **Improved Netlist Generated for ProbeMaster**

Better handling of soldermasks opening pads the center of which is slightly offset from the center of the copper pad they expose. This works only with ProbeMaster V11.2.1 or higher.

11.12.18

- **Larger Toolbar Buttons**

Many of the toolbars have buttons that are difficult to see on high resolution monitors. There are now new buttons in many of the toolbars and their size can be picked by the user ("View/Toolbars/Button Size" or "Setup/Preferences/Toolbar Button Size").

11.12.16

- **NC Tool Numbers Range**

When importing NC drill and rout data, tool numbers up to T999999 are now accepted. Internally the allowed ranged is still T1 through T255, so tool numbers larger than 255 will be transcoded during import.

- **Pocket Routing**

Added option for routing in vertical direction.

11.12.14

- **New Tool for Organizing Layers**

Can be activated from several places, including "Setup/Layer Organizer" and an icon on the layers toolbar. It makes it easier to organize and identify the individual layers of the loaded data. Pressing the help icon in this add-on gives a complete description of its capabilities.

- **Pocket Routing**

"Tools/Routing/Pocket Routing" converts selected polygons into routing paths that can be used to cut out the area the polygons represent while not cutting through the whole material (this generates pockets).

- **Import .pdf**

"File/Import/PDF As Vector" now imports separate pages of the PDF file into separate layers.

- **3D Viewer Export to .stl Files**

The 3D Viewer can now export .stl (stereo-lithography) file of the image. One file per image entity is exported because the .stl files do not contain color info.

- **New shortcut: Ctrl+Shift+F5**

This is for displaying the NC tools setup dialog. There are two version of it: the legacy and the newer version. One can pick which one is displayed with "Setup/Preferences/Use Old Tool Codes Dialog". Pressing "Shift+F5" will display the one picked by this preference setting. Pressing "Ctrl+Shift+F5" (new shortcut) will display the other one.

11.12.9

- **Visualizing Scratch Elements**

At times it is hard to see where scratch elements exists on a layer. There is now a quick way to select them in order to see where they are: "Select/Special/Scratch". This works only on active elements, the same way as the selection commands in "Select/Special".

- **3D Viewer Improvements**

11.12.6

- **3D View of Boards**

The menu bar has a new item called "3D View".

This allows starting a 3D viewer of the currently loaded board data. The separate application is a viewer only, the board can be viewed but not modified.

The image data is generated in CAMMaster and relies on layers being correctly labeled with layer type.

The following layer type will be used to generate the image:

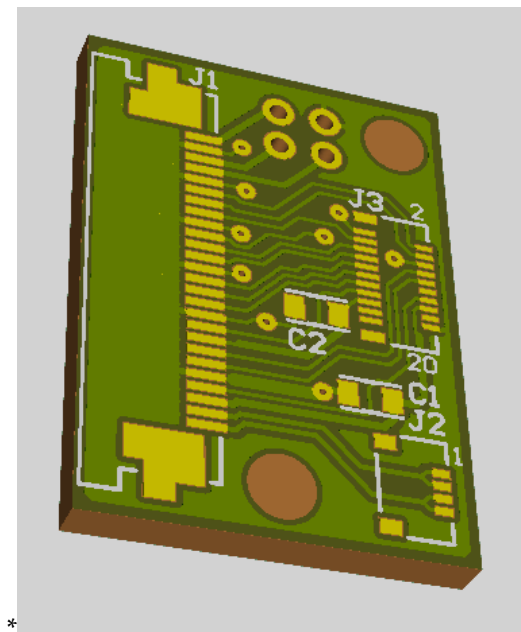
BOL, CPU, CPL, SKU, SKL, MSU, MSL and DRI.

At least BOL and CPU must be present in order for an image to be generated.

The BOL layer (board outline) must contain a valid board outline. Either a set of connected traces or one polygon.

The viewer uses OpenGL to render. Because of this, it will only work on graphics adapters that support OpenGL and preferably V6.14 or later. Usually updating the graphics card with the latest driver from the card manufacturer or computer manufacturer takes care of this problem.

A sample view of a simple board is shown below:



11.12.1

- **Toolbars Positioning**

Toolbar positions and docking are now remembered separately for different screen resolutions. This helps users with laptops and docking stations, where resolution can change when docked.

Toolbars must be arranged again when resolution changes, but once set positions will be remembered. Another change is that toolbar positions will not need to be reset when a new major version of the software is released.

- **New Stencil Shape**

V shape:



11.10.80

- **Convert a Single Drawn Pad**

“Tools/Convert Drawn Pads/Convert Selection to One Pad”. This new command will take all selected data and make a pad out of it (most often a custom shape, “M”). This is useful when a pad is needed at that location and the copper area of the item is not connected (made of several fragments).

11.10.77

- **New DFM Report Viewer**

New module that allows viewing of DFM reports generated by the PCB Analyzer product, using “live CAM data”. “Tools/EMS Magician/DFM Report Viewer”. For more detailed release notes please read the viewer release notes found in the installation folder.

- **Export Routing Path: Finished Routing Function Support**

Excellon M16, Sieb & Meyer G11.

- **Convert Circles to Pads**

Some CAD output contains circles (arcs) where pads are intended. This new operation converts them to pads. “Edit/Edit Selection/Special/Convert Circles to Pads”. Only full-circle arcs will be converted.

11.10.73

- **Import ODB++**

Support for dynamic text: \$\$JOB, \$\$STEP and \$\$LAYER.

- **Save and Restore DFM parameters**

One can now save sets of parameters to text files. “Custom DFM”: “Save Parameters To File” and “Restore Parameters From File”.

- **Select Approximate Duplicates**

A new method to select duplicates that are not identical. They can differ by a user specified tolerance. “Select/Special/Approx Duplicate Elements”.

- **A Strict Method for Finding Routing Paths**
"Tools/Routing/Strict Group to Paths". This will only continue a path where endpoints meet exactly. No intersection or touching is considered.
- **Contour Netlist Extraction**
Endpoint / midpoint computation has been improved.

11.10.67

- **Transcode from "N" shapes to "C" shapes**
At "Tools/Holes to Pads". This is the reverse of the already existing "Pads to Holes".
- **IPC-356 from ODB++ file**
When extracting an IPC-356 netlist file from ODB data ("Tools/Netlist/Convert Netlist File/To IPC") reference designators are also generated in the IPC-356 file.
- **Vertical Adjacency in Contour Netlist**
An option has been added to compute vertical adjacency also when doing contour netlist.
Vertical Layer Adjacency: Identified nets from the layers above and below the specified net to be included in the isolation test. Each net from one layer above and one layer below is considered as if it was on the same layer as the specified net and is included in the vertical layer adjacency list if it lies within the vertical layer adjacency distance. This distance can be different than the horizontal layer adjacency distance.

Tester Options

☐ Generate Phase Test Data

☒ Generate Net Adjacency Data

☐ Vertical Adjacency Also

☐ No need for shielded adjacency

☐ Shield Adjacency (Ignore Blobs, Recommended)

☐ Shielded Adjacency (All)

50 Minimum Net Separation (mils)

50 Separation (mils)

11.10.65

- **ODB++ Export**
Step and repeat implemented.

11.10.63

- **Gerber X2 - TF.FileFunction Implemented**
The FileFunction attribute for Gerber X2 has been implemented for export and import. When exporting a file set in X2 and importing it using Import/Directory or Import/Zip, the proper layer order is preserved.
- **Gerber X2 – Nested Step and Repeat Implemented**
Both import and export.

11.10.56

- **X and Y Options Added for Drill Serpentine Sort**
In “Tools/Drill”. If X is picked the sort will go on the X direction first.

11.10.54

- **Preliminary Support for Exporting Gerber Extended V2**
Option added for V2 export. More exported items will be added later.
- **Improved ODB++ Export**
Custom D Codes and routing paths are now exported.

11.10.50

- **Import of PDF files**
“File/Import/PDF”. Note that only pdf files that have vector content, such as fab drawings, will give good results. One such example is in the Samples folder.

11.10.44

- **Sax Basic replaced with WinWrap Basic**
The IDE for Basic scripting was switched from Sax Basic to WinWrap Basic. Sax Basic is no longer supported and WinWrap Basic is a more modern tool. All previously written scripts should work unmodified.

11.10.43

- **Export ODB++**
The whole loaded data is exported to a ODB++ file. Some features, like routing paths and step and repeat, are not yet implemented. Will be added in upcoming versions.

11.10.40

- **Built with Visual Studio 2013. Does not need .Net V2.0 any longer.**

11.10.24

- **This is the last release built with Visual Studio 2008**

11.10.17

- **New option for exporting rout files**
The “Rout Options” tab has a new item “G in every block”. When checked the exported file will contain a G field in every block. Apparently some router need this.
- **Tool Codes Setup dialog improvements**
 - The “Swell Drills” operation allows swelling by positive or negative values.
 - A new operation “Merge Tools” allows merging all the tools that have the same type and diameter into one tool. This applies to drill holes as well as routing paths.

11.10.14

- **New selection command for selecting small elements**
“Select/Special/Small Area”. Will select element whose area is smaller than the value specified for the user. This is useful for finding very small elements hidden by other elements or just hard to see because of their size

11.10.9

- Improved computation of midpoints when generating a contour netlist.
- Faster User Interface for the Tool Codes Setup dialog
- Improved License Activation Dialog

11.10.1

- Improved parsing of Gerber files with modal flashes (D03).

11.8.24

- Improved computation of midpoints when generating a contour netlist.
- **Improved parsing of Gerber files with modal flashes (D03).**

11.8.1

- Improved polygon swelling algorithm

11.6.41

- **New Tool Codes Setup dialog (Shift+F5)**
With this release, in addition to tool diameter and tool type, you can now set the following drill and rout tool parameters:
 - ✓ FEED – This is the “InFeed” of the tool and determines how quickly the tool is plunged into the material. Units are either “in/min” or “m/min”.
 - ✓ SPEED – This is the tool rotation speed in K-RPM. A value of 100 would mean 100,000 revolutions per minute.
 - ✓ RETRACT – This is how fast the tool is removed from the material at the end of a cut or a drilling cycle. Units are either “in/min” or “m/min”.
 - ✓ ZOFFSET – This determines how far the tool is plunged into the material (relative to a reference plane). Units are either “in” or “mm”.
 - ✓ TABLEFEED – This is how fast the rout tool is moved along the contour of the rout path. Note that this is an optional parameter and only applies for rout tools, not drill tools.
 - ✓ MAXHITS – Determines how many holes can be drilled with a particular tool before requiring a new tool.

You can save multiple feed and speed profiles and recall them for later use. This allows you to use different profiles based upon layer count, material type, etc. You can also use the built-in Column Organizer to import existing Feed and Speed files (commonly in Excel .csv format).

If your drill file does not contain feed and speed information, you can map the feed and speed parameters by utilizing the loaded "Feed and Speed Profile". By clicking on the "Find Closest" button, we will search the loaded profile and find the closest tool. The new diameter and all associated feed and speed parameters will be shown in the Tool Codes menu. You can setup your feed and speed profiles to contain only the tool diameters that you stock. This way, you can be assured that a selected tool will be one that you have available.

In addition, you can now automatically swell drill sizes to compensate for drill etch factor. You can set multiple drill etch factor ranges based upon tool diameter.

For comprehensive information about how to use the new Tool Codes menu, you can click on the "Help" button after you launch the new Tool Codes dialog (Shift+F5).

If you prefer to use the old, legacy Tool Setup, you can do that by clicking on "Setup/Preferences/Use Old Tool Codes Dialog".

- **New Excellon export option**

It is called "Omit m.n" and it can be used to omit the output of the "00.0000" type m.n format in the INCH/METRIC line. This is a newer Excellon format spec and is not recognized by all parsers.

- **New Sieb & Meyer export option**

It is called "Rout tool diameters as negative numbers". When checked, the diameter for tools that are of type "Rout" will be exported as negative numbers.

11.6.35

- **IPC-356 test points imported**

When importing netlist data, "099" records, if present, will be imported as pads on separate layers. These records were previously ignored.

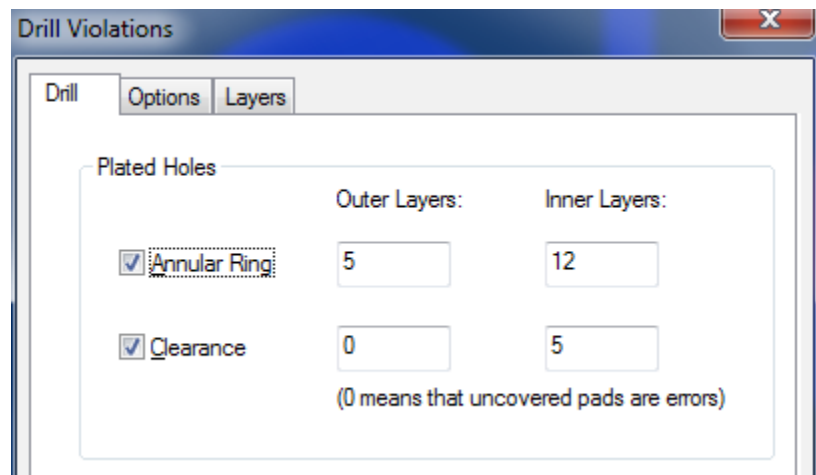
11.6.26

- **Improved Stencil Pattern Search**

When searching for a model a tolerance of 0.1 mils (or 1 micron in metric) is used. This allows for finding components that are slightly off due to rounding errors.

- **New DFM Check For Uncovered Plated Holes**

For "DFM/Drill Violations", if one sets the "Clearance" value for plated holes to 0, then plated holes that are not covered by copper will be flagged as errors:



- **NC Rout Path Plunge**

The existing command "Set Plunge at Vertex" has been renamed "Set Plunge" and now allows the plunge point to be set anywhere on the path, not only at vertices.

- **New Command to Adjust a Soldermask Layer**

"Tools/Soldermask/Adjust Existing Soldermask".

This will take an existing soldermask layer and adjust it for purposes of netlist extraction and electrical test output. It is most useful for netlists to be used with PentaLogix ProbeMaster.

It takes any visible MSU and MSL layers and checks for needed adjustments. If adjustments are made, new soldermask layers are generated. The original ones are left unchanged.

The following adjustments are made:

- If a soldermask pad does not have the same center as the pad it covers, the center will be adjusted to match
- If a soldermask pad covers more than one component pad, each component pad will get its own soldermask pad
- If a component pad is covered by a soldermask area that is not a pad (perhaps drawn traces or polygon), a new soldermask pad will be added
- Existing soldermask pads that do not cover any component layer pads are discarded
- Pads in the component layer that are not covered by soldermask will get a new soldermask pad. These added soldermask pads will go into a separate adjustment layer. The reason for this is that not all uncovered component pads need soldermask. It is up to the operator to inspect this layer and decide which new soldermask pads are needed and which are not.

The resulting adjusted soldermask layers will contain only pads.

Note that in the above discussion wherever the term covered by soldermask was used it means that there is a soldermask opening on the board. Soldermask layers are usually negatives but the discussion is in terms of the CAM pads.

If a soldermask layer has been adjusted, any newly generated pads or moved pads will be shown as selected on the new layer to make it easier to see the changes.

11.6.18

- **Convert Obround Pads to Traces**

This command will convert all O-type pads (obrounds) to geometrically equivalent traces. "Edit/Edit Selection/Special/Convert O-type Pads to Traces". This is useful if the destination system does not accept arbitrary shaped pads or when one wants to export an N/C file. The traces will be exported as nibbles, whereas obround pads cannot be exported as N/C.

- **Stencil Aperture: Multiple Models**

One can apply multiple stencil models on the selection with the new command "Tools/Stencil Apertures/Convert From Library".

- **Two Types of Stencil Windows Models**

One can now generate windows-type models that apply to all selected rectangular pads or just to pad of a given shape.

- To generate a model that applies to only one shape, select only one rectangular pad. Do "Tools/Stencil Apertures/Windows". The model created this way will apply only to pads of the same dimensions as the selected pad.

- If more than one pad was selected when the windows operations is performed, then the resulting model will apply to any rectangular pad.

11.6.11

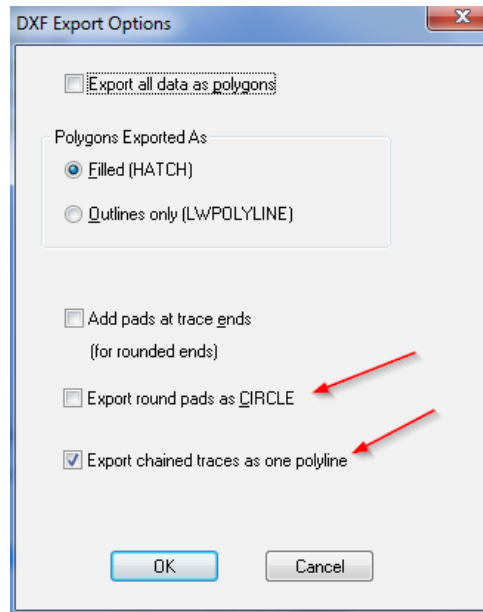
- **Stencil Aperture Libraries**

The stencil module has been improved to allow for saving and reusing aperture models. This is described in an application note:

[v](#)

- **Improved DXF Export**

Two new export options have been added:



"Export round pads as CIRCLE" will cause round pads to be exported using the DXF CIRCLE entity. This shows in AutoCAD as a non-filled circle, but it is desirable in certain cases.

"Export chained traces as one polyline" will cause a chain of traces to be exported as one polyline. If unchecked, each trace will be exported as a separate polyline.

Chains are groups of traces where one trace's endpoint is the same as the next trace's starting point and all traces have the same width. Export as chains makes editing in AutoCAD easier and also results in a considerably smaller .dxf file.

11.6.5

- **Improved Thermal DFM**

"DFM/Thermal Violations": the algorithm for finding violations has been improved.

11.6.0

- **Easier access to endpoint alignment option**

When aligning or selecting traces and polygons, there is an option to move the cursor to the endpoint only. Since this is a frequently needed operation, a new button has been added to the Selection Toolbar and a new entry in the context menu (mouse right-click).

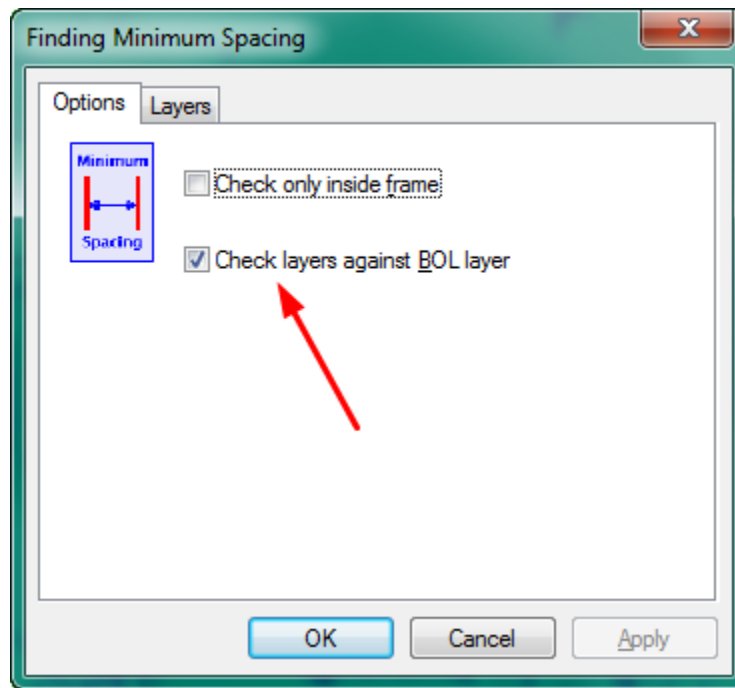
- **New button to pan up**

A new button has been added in the "Screen" toolbar to supplement the "Next Screen" button. The new button is "Next Screen Up" and it will step in the Y direction, rather than in the X.

11.4.45

- **Minimum Spacing to Board Outline**

"DFM/Find Minimum Spacing" has an option to also compute the spacing to the board outline. There must be a separate layer of type BOL that contains the traces for the board outline.



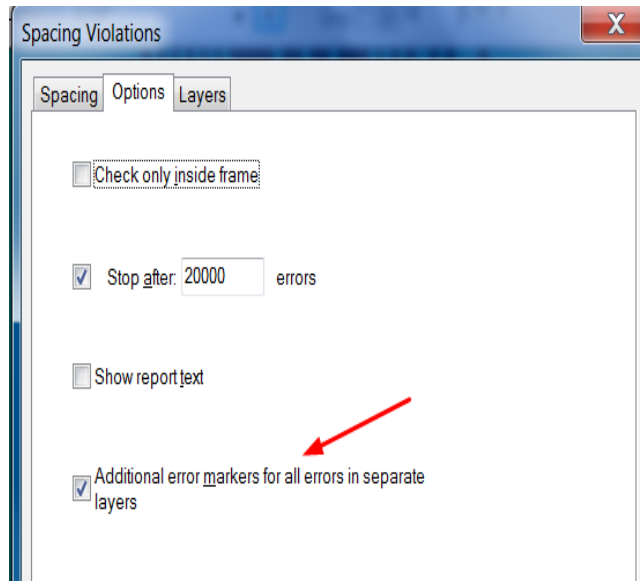
11.4.42

- **New Layer Type**

New layer type for stencil layers have been added: STU (Stencil Upper) and STL (Stencil Lower).

- **Error Markers for DFM Checks**

Spacing and width violations now have a new option ("DFM/Spacing Violations", "DFM/Feature Size Violations"). When turned on, it will generate (in addition to the markers for viewing) zero-width traces in empty layer for each of the markers. This way data can be saved with the error markers on separate layers for later viewing.



11.4.37

- **Gerber Export: New Option for Polygons**

One can choose to export polygons as filled with traces (of a user picked D Code).

- **DXF Export: New Option**

A new option has been added ("Add pads at trace ends") that controls if round pads are added at the ends of traces mapped to round D Codes. If checked, the round pads will be added to make the DXF drawing look like the Gerber image. Otherwise the rounded ends will not be added. This is sometimes useful when the output is used for routing or other tools where the rounding does not matter and may even interfere.

- **Stencil: New Options**

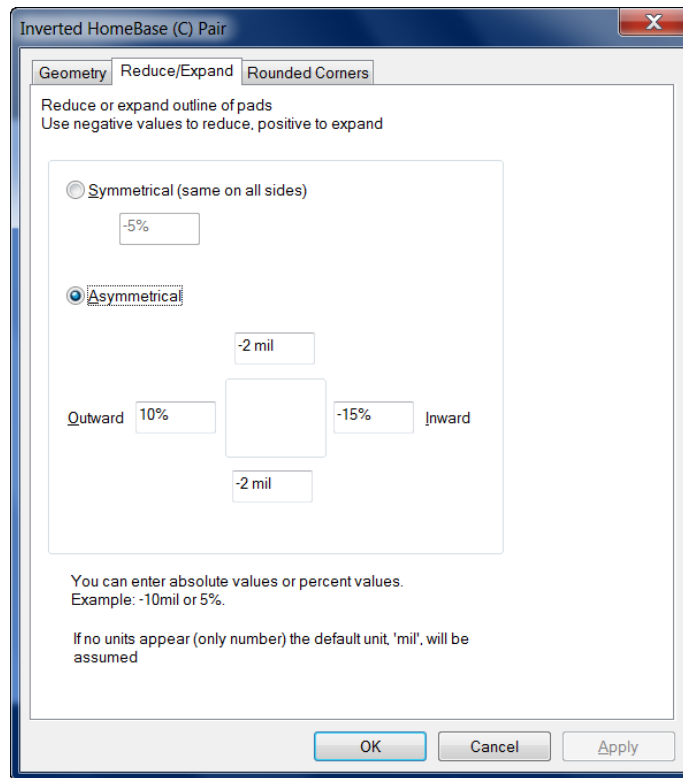
⇒ The "Windows" mode for stencil generation has new options for diagonal splits:

- crosshatch (this was the only mode before)
- ascending
- descending.

⇒ The inverted homebase has a new geometry rule for rounded (called "Simple": straight feet and rounded center). This is only active when "Rounded" is also checked.



⇒ One can now mix parameters that are given as absolute values and percent, by explicitly entering a percent sign or units. If neither is given, absolute will be assumed in the current units. An example is shown below with mil units for certain parameters and percent values for others.



- **Spacing and Width DFM**

Improvements have been made to the spacing and width violations DFM modules.

11.4.29

- **Option for Transcoding Report**

During import (Gerber and others) D Code numbers may need to be transcoded to preserve the specified shape of the D Code. During transcoding a report file is generated. This added option has three choices (available under "File/Import"):



One can:

- ⇒ disable report to file ("None")
- ⇒ generate a file called "Transcode Report.txt" in the same folder where the imported file is ("To Import Folder")
- ⇒ use the file that was picked previously with "File/Report/Set Name" ("In Report File").

- **Gerber Export User Interface**

The two menu entries for exporting Gerber ("Basic Gerber" and "Gerber Extended") have been combined into one. The dialect of Gerber (Extended/Basic) can still be picked from the "Options" dialog.

This change has been made because exporting Basic Gerber is only needed in very rare occasions. Extended Gerber (RS-274X) is almost always the preferred output, since it contains format and shape info.

- **Easier Format Guessing**

The dialog that is shown when one does "File/Import/Guess" (F4) now includes the board size limits to allow for changing them and trying to guess again. Previously one could only set the board size limits on which the guess was based in the "File/Import/Directory/Options" dialog.

- **Turn Off Full Path in Print Header**

A new option has been added to the print options called "Full path". If unchecked, only the filename part of job names or layer names will be printed in the header.

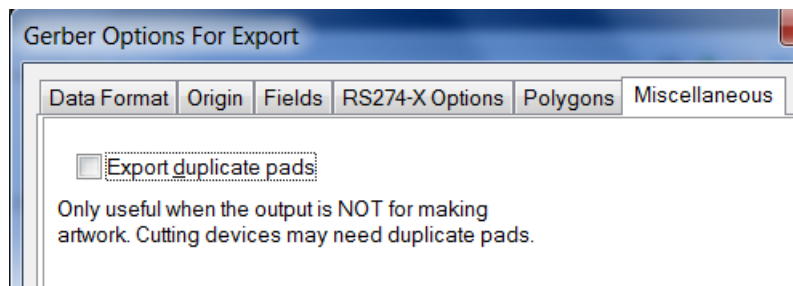
11.4.21

- **Stencil Aperture Generation Improvements**

Obround shapes can now be inflated using the "asymmetric" mode.

- **Option to Export Duplicate Pads**

An option has been added for exporting Gerber files with duplicate pads. These are two or more pads at the same location in the same layer and of the same D Code. Exporting more than one makes no sense when producing circuit artwork, but it is sometime needed for sending the data to certain other NC machines. Unless you specifically need it, leave this option unchecked.



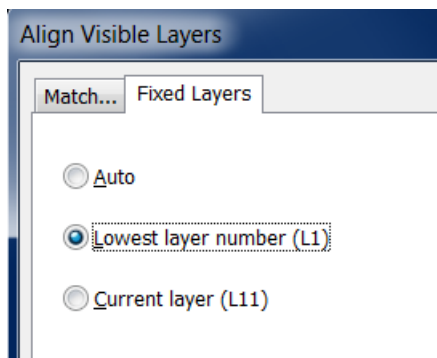
11.4.15

- **ODB++ V7.0 Support**

Rotation of pads (by arbitrary angles) in ODB++ files (introduced in V7.0 of ODB++) is now supported.

- **New Option for Aligning Layers**

"Edit/Move Layers XY Position/Align Visible Layers" (Ctrl+Q) now has a new option tab for allowing the user to select which layers will be moved and which will be left standing during the alignment. There are 3 possibilities:



- 1) "Auto" will work as this command worked before. The software will determine which layers to

- move and which to keep in place.
- 2) "Lowest layer..." will keep in place the lowest layer number among the visible layers.
- 3) "Current layer" will keep in place the current layer.

11.4.10

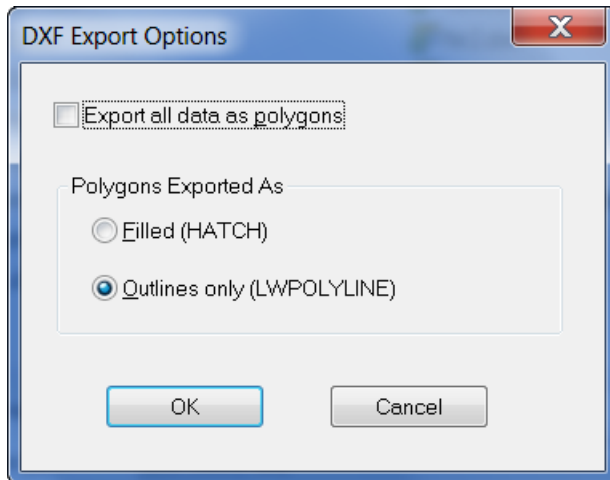
- **Rotated Rectangles for Flash Contours**

"Tools/Convert Drawn Pads/Flash Contours in Selection" now converts rotated rectangles as well. Previously only orthogonal rectangles were detected and converted.

- **DXF Export Options**

An "Options" button has been added to the export DXF dialog.

It is shown below:



If "Export all data as polygons" is checked, then all the data (traces, pads etc.) will be converted internally to polygons and then exported as polygons. This usually results in smaller data files, but they may be harder to edit in AutoCad.

The other option controls how the polygons are exported (this applies whether the "Export all data as polygons" was checked or not). The options are filled (using the DXF HATCH entities) or outlined (using DXF LWPOLYLINE entities)

11.4.6

- **Import DXF**

A new options dialog has been added: "File/Import DXF Options". It currently has settings for specifying the units for the DXF file. Previously, the current units in the CAM software were used.

- **Export DXF**

Layers that do not contain scratch elements and are not negative are now exported without converting everything to HATCH entities. POLYLINE and text entities will be used where possible.

- **.bin File Open by Double-Clicking**

A new command has been added to associate or disassociate the ".bin" file extension with this software. It is "Setup/Advanced/Register for .bin files". When checked, one can double-click on a .bin file in Windows Explorer to cause it to open with this software. When unchecked CAMMaster does not associate itself with the .bin extension.

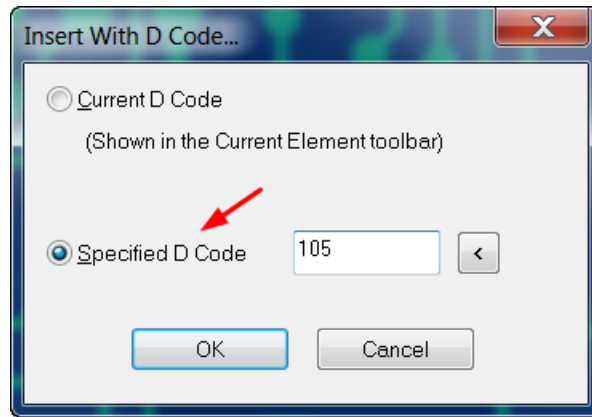
- **New Draw to Flash Operation**
"Tools/Convert Drawn Pads/Flash Contours in Selection" will replace contours drawn with F-type D Codes with pads. Currently only round and rectangular shapes are supported .
- **Layer Info for Printing**
"File/Print" has a new option for adding a line with layer info (layer name, type, etc): "Add layer info line".
- **Improved Feature Size Violations**
The algorithm for finding width violations ("DFM/Feature Size Violations") has been improved.

11.4.0

- **Import DXF**
One can now import AutoCAD DXF files ("File/Import/DXF").
Some features in DXF are not yet fully supported, but the current implementation should be sufficient to allow import of most PCB related DXF data.
DXF files can contain text in a variety of fonts and our CAM software will map them to the PentaLogix built-in fonts. Because of this, sometimes text spacing will look different than in the original. SPLINE entities are currently not supported.
More support for unimplemented features will be added in upcoming releases. Please send us .dxf files that do not import correctly for you.
- **Single Point Net Numbers Listed in Net Count Report**
When extracting netlist using the contour method and "Generate Net Count Report" is checked, the net numbers of single point nets are listed at the end of the file in addition to the count of such nets (which was the only info generated before in the report file).
- **Contour Netlist and Obround Shapes**
Obrounds are not supported in the IPC-356 format. So, when the CAM data contains obrounds they need to be converted to rectangles or rounds. The computation of when to use rounds instead of rectangles has been improved.
- **Rout Export: Multi-quadrant Arc Option**
A new option has been added when exporting routing paths in the "Options" dialog in the "Rout Options" tab. It is called "Quadrant arcs only" and will control whether arcs that span several quadrant will be exported as one arc or several arcs (one per quadrant). The latter was the default mode before, but there are more routers now that can accept multi-quadrant arcs.

11.2.78

- **Picking D Code for Inserting Elements**
A new user selection has been added to allow picking a specific D Code for inserting elements (pads, traces etc.). Previously newly inserted elements were using the current D Code (from the Current Element toolbar). That is still the default, but the user can now pick a specific D Code for inserting elements.



This is useful when during insert operations the current D Code can change (due to aligning or selection) but the user wants to insert using a specified D Code.

The command to pick this option is under "Insert/Using D Code". This is similar to the existing "Insert/Into Layers".

- **Screen Toolbar Split in Two**

The Screen toolbar has been split into two toolbars, now called **Position** and **Screen**. This is to allow users the choice to turn off one or the other independently. The Position toolbar contains the units, X and Y coordinates and zoom, which are the items more frequently used. Users who don't need the other buttons can now turn the Screen toolbar off and have more room on the screen.

- **Gerber Coordinates with Decimal Point**

"File/Import/Directory" has been improved to recognize Gerber files with coordinates with decimal point notation. This is not standard Gerber, but such files do exist.

- **New Option for Gerber Import**

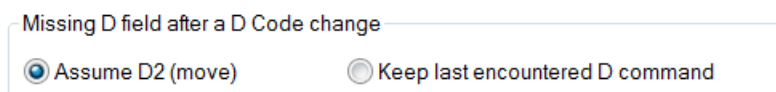
The new option is in the "File Interpretation" tab of the options dialog and controls the handling of Gerber blocks without a D command following a tool change (D Code change). An example is shown below:

```
X163830Y76835D01*
D24*
X161290Y75565*
X157480Y76200*
```

The question is how to interpret the block: X161290Y75565*

According to the Gerber spec this should be a move, but some software outputs it expecting that the D command in the previous block should be preserved (D01).

Because of this a new option has been added:



It is safest if Gerber files contain a drawing D command in every block (D01, D02 or D03) but this is not always the case.

- **Improved Midpoint Computation**

The algorithm for computing which test points are endpoints and which are midpoints has been improved in contour netlist extraction.

11.2.74

- **Option for "LN" Directive**

A new option has been added to enable or disable the output of the "LN" (Layer Name) directive in Gerber Extended. It is in the "RS274-X" tab of the export "Options" dialog. Previously the "LN" was written for any layer that was named.

- **Trim Traces Improvement**

"Edit /Trim Selected Traces" now has an additional parameter for clipping. It is a percentage number that allows a user to set the maximum percent of a trace that will be clipped. A maximum length of the clip parameter existed before and it is still present. Only if both are satisfied will a trace be clipped.

- **Improved Plane Nets Classification**

Contour netlist has a user parameter called "Minimum Plane Size". Users can set this value to cause certain nets to be categorized as "plane nets" (net number less than 200). Previously the value was used to compare against the dimensions of the bounding box of the copper. Now, the copper area of the net is used for this computation. The area is compared against the square of the value entered and, if larger, the net is considered a "plane net".

11.2.71

- **Check For Updates**

"Help/Check For Updates" will now open in a web browser instead of a child dialog of the application. This allows the application to be closed while the update is being done.

- **Import Directory Improvements**

Better guess for format of drill files.

11.2.64

- **DXF Export**

"File/Export/Autodesk DXF" will export chosen layers in DXF format. One can export layer by layer or combined. Layer by layer will generate one .dxf file for each chosen layer, the file name derived from the layer name. Combined will export all picked layers into one file, but this is of limited utility when several layers are chosen, because the colors are not translucent in DXF. However, this is useful when exporting just one layer when one wants to specify the file name during export.

11.2.60

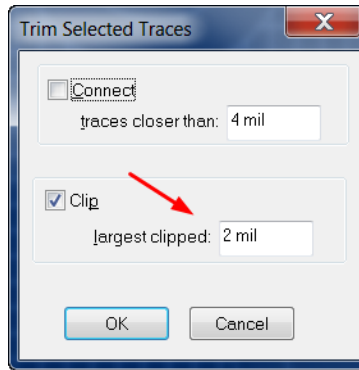
- **Easier Transcoding**

Transcoding groups of D Codes from the "Setup/D Codes" dialog (F5) has been changed ("Operations/Transcode"). The destination range (range of D Code numbers to which the existing D Codes are transcode) does not need to contain a contiguous range of unused D Codes large enough to accommodate the source range. If enough unused D Codes exist in the destination range, the operation will complete. Previously the operation would fail if there was no contiguous range available.

11.2.58:

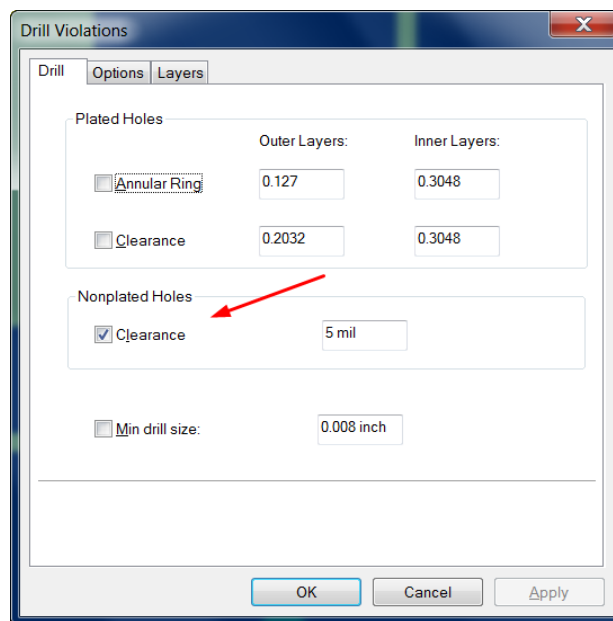
- **New Parameter for Timming Traces**

A new parameter has been introduced for clipping selected traces. This specifies a maximum length. If the clipped portion would exceed this length the trace is not clipped:



- **New DFM Check for Unplated Holes**

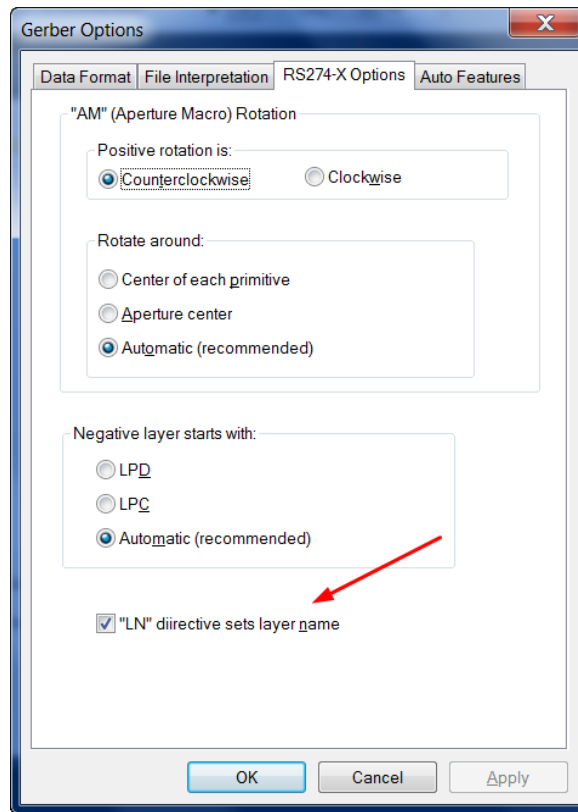
The drill DFM module has a new check for unplated holes:



If enabled, it will find non-plated holes that are closer than "Clearance" to copper on any of the conductive layers and will also flag as errors non-plated holes that go through copper without a clearance.

- **Support for Gerber Extended "LN" Directive**

A new option has been added for importing Gerber Extended data :



When the "LN" option is checked, the string that appears in the "LN" directive (Layer Name) in Gerber Extended files will set the name of the layer in the CAM software.

11.2.52:

- **New Gerber Export Option**

A new option has been added to the Gerber export dialog. This is to control the data output at the end of a plot. Traditionally a plot ends with "X0Y0D02*M02*", but certain devices have problems with the "X0Y0D02" part. An option has been added to specify if it should be output or not. It is in the "Fields" tab of the options dialog and it is called "X0Y0D02 before end of plot".

- **Selecting Traces By Aperture Size**

"Select/Special/Wide Traces" and "Select/Special/Narrow Traces" have been added. This works similar to the existing "Large Pads" and "Small Pads" commands. They will select traces based on the diameter of their D Code. Selection is only among active elements (taking into account the other filters, if any).

11.2.45:

- **Clip to Board Outline**

A new executable has been added. It is called "CAMMasterSampleScripts.exe" and it is located in the "Scripts" subfolder. To find out more about what it does click on the "Help" menu after starting it. It currently provides only one script, which clips data on a layer to the board outline. In the future more such useful script modules will be provided. Please contact PentaLogix if you have a specific need for an operation that can be scripted.

- **New Netlist Export Options for ProbeMaster**

The PentaLogix electrical test software (ProbeMaster and FixMaster) currently only allows for round pads

as hole covering pads. If the pads are rectangular they will be processed in the ETest software as SMD pads. Sometimes this is not desirable (for instance when one wants to stagger the pads after export to the flying probe tester - some tester can stagger pads, but need covering pads for this). To help with this, a new option has been added to the contour netlist export dialog in the "padstack" group: "Round cover". This will export holes covered by rectangular pads as if the pads were round (inscribed in the rectangle).

11.2.43:

- **Printing on White Background**

The "Screen colors on white" has been improved to print the exact colors visible on the screen. The "Use Raster method when printing" checkmark must be checked for this to work. This is useful when one wants the printed image to have the same colors as the screen but one wants to save ink and not print the black background.

- **Selecting Small Pads**

A command has been added for selecting active pads mapped to D Codes with at least one dimension smaller than a user specified value. The command is "Select/Special/Small Pads". It takes into account the selection filters as well, except for the D Code filter (it always checks all D Codes). If for instance one wants to select only small round pads, one can pick the "Shape C" filter item in the Selection Toolbar.

- **Polygons Filter Shortcut**

A new shortcut has been added for using polygons as the selection filter: "Ctrl+Shift+Y". This applies to the filter box in the Selection Toolbar.

- **New Selection Filters**

The filter combobox that appears in the Selection Toolbar (and other dialogs) now has several new added items: "Plated Holes", "Non-plated Holes" and "Via Holes". This restricts selection and operations that work on active elements to just pads of type "N" and mapped to holes of the specified type.

- **D Code Layer Usage Report**

"File/Report/Generate" will now also show on what layers each D Codes is used.

11.2.35:

- **New Midpoint Option for Netlist Export**

A new option has been added to contour netlist file export. It is called "Padstack midpoint". When checked all pads in a padstack (the hole and any covering pads) will be assigned the same midpoint / endpoint state. If the netlist computation determined that any of them is an endpoint than all will be set as endpoints, otherwise all will be set as midpoints.

Some testers, like the ATG testers, seem to prefer this mode.

11.2.31:

- **Turn On and Off a Range of Layers (Visibility)**

A new way of changing the visibility of a range of layers has been added to the layers toolbar. To turn ON a range of layers click on the first and Shift+double-click on the last in the range. Similarly to turn OFF a range of layers click on the first and Ctrl+double-click on the last.

- **Stretch Traces Orthogonally**

A new option has been added to allow only orthogonal (horizontal, vertical or at 45 degrees) stretch of straight traces: "Edit/Orthogonal Stretch Only".

- **Paint and Scratch Optimize for Layer**

A command has been added to the right-click menu in the layers toolbar: "Paint and Scratch/Optimize". This will attempt to minimize the number of paint to scratch transitions on a layer by merging groups that do not interfere and adding scratch polygons as holes to containing paint polygons.

- **Guessing Layer Types**

A command has been added to the right-click menu in the Layers Setup (F10) dialog: "Guess layer types from name". It will attempt to determine the type of a layer ("CPU", "CPL" etc.) from the layer name and extension using CAD software standard naming conventions.

- **Select Dangling Traces**

A new command has been added for selecting dangling traces ("Select/Special/Dangling Traces"). This command will look only at active traces, so all the filters and "Only" check marks are taken into account. A dangling trace is a trace which has at least one uncovered endpoint. Normally trace endpoints are covered either by a pad or another trace. If that is not the case, it is usually a problem on circuit layers. One can specify areas to exclude from the check using exclusion zones ("Setup/Exclusion Zones/Insert").

- **Break Intersecting Traces**

This new command ("Edit/Break Intersecting Traces") works on the selected traces (linear or arc) and breaks traces that intersect at points other than the endpoints of the traces. Such traces are replaced with the resulting fragments.

- **Exclusion Zones for DFM**

One can now specify a set of rectangular regions (exclusion zones) when running DFM. The area covered by these exclusion zones will not be processed during DFM. This is useful for avoiding the spurious errors that can be caused by text or targets, etc. on copper areas, but it can be used for excluding from the DFM check any unwanted areas. The exclusion zones can be manipulated using "DFM/Exclusion Zones".

- **Spacing Violations Improvements (DFM)**

A new option has been added for checking spacing violations between parts of a connected copper region (same net - "Any"). This checks for spacing violations at all points of the region and can be run instead of the acid traps check.

The check between separate copper regions will now report all violations (previously only one was reported per pair of regions).

11.2.26:

- **Distance to outside edges**

The edge-to-edge measurement tool (right mouse click - "Edge To Edge Measurement Tool") now displays the distance to the outside edges of the two picked elements, in addition to the distance to the inside edges.

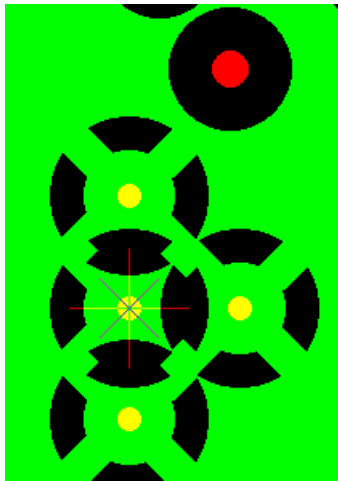
- **Dimensions shown on screen**

When running one of the "DFM" tools the results displayed on screen include the numerical value of the measurement in addition to the arrow.

- **Starved Thermals DFM check**

A new check has been added to the "DFM/All Violations" module. It checks for starved thermals (thermals that do not have a wide-enough connecting path to the rest of the plane). For this check one needs to pick the circuit layers to be checked and the corresponding drill, via or soldermask layers. Only locations on the circuit layers that have a crossing drill hole or a soldermask opening are checked. Problems like

the one shown below are reported by this check:



11.2.22:

- **View Layers where a D Code appears**

A new command has been added to the "Setup/D Codes" (F5) dialog that will display the list of layers that contain the D Codes currently picked in the dialog. The name of the new menu command is "Layers".

- **Faster Contour Netlist**

The algorithm speed has been improved for jobs with many blind and buried vias.

11.2.18:

- **Drill Size DFM**

A new DFM check has been added to the drill DFM rules ("DFM/All Violations/Drill"). It checks for holes of diameter smaller than a user specified size.

11.2.8:

- **Excellon Format 1 Import**

An option has been added to the "Options" dialog for "File/Import/Drill & Rout" to allow setting either Format 1 or Format 2 as the format of the data to be imported. This is important when the Excellon file does not contain a FMAT statement because Format 1 and Format 2 commands have different meanings. Previously Format 2 was assumed as the default.

11.2.1:

- **Editing Layer Names**

New commands have been added to the context menu (mouse right-click) in the "Setup/Layers" dialog (F10): "Load layer names from file" and "Store layer names to file". These allow saving the layer names to an ASCII text file and also loading the layer names from a file. This can be useful in several ways:

- ⇒ To load the names from a fixed template of layer names
- ⇒ To edit the names when a large number of names need to be changed. To do this, one can do the following: save the current names to a temporary text file, edit the file with notepad or other text editor that allows for group replace operations, finally reload the names from the just edited file.

- **Converting Pads to NC Holes**

There are cases where there is no input drill file, but there is a Gerber or other circuit data pad master. A new tool has been added to convert such layers to NC layers that can be exported as drill data. The new command is "Tools/Pads To Holes".

In the layers picked by the users it will transcode any elements mapped to a D Code of shape "C" to a new D Code of type "N" (NC hole). These elements can later be used to generate drill files, to extract netlists or to use in Smart DFM. This command transcodes traces as well, although that is generally less useful. Only visible elements will be transcoded.

11.0.80:

- **Improved Copy and Paste**

The "Edit/Copy" (Ctrl+C) and "Edit/Paste" (Ctrl+V) commands now use as reference point the position of the cursor when the copy command is performed. Previously the lower-left corner of the bounding box of the selected elements was the reference point.

- **Break Tabs at Corners**

A new command has been added to automatically insert 4 break tabs at the corners of closed routing paths. This is intended to be used with rectangular routing path, but will work with any shape. The command is "Tools/Routing/Break Tabs at Corners for Selected". It will change only selected routing paths and only the ones that are closed.

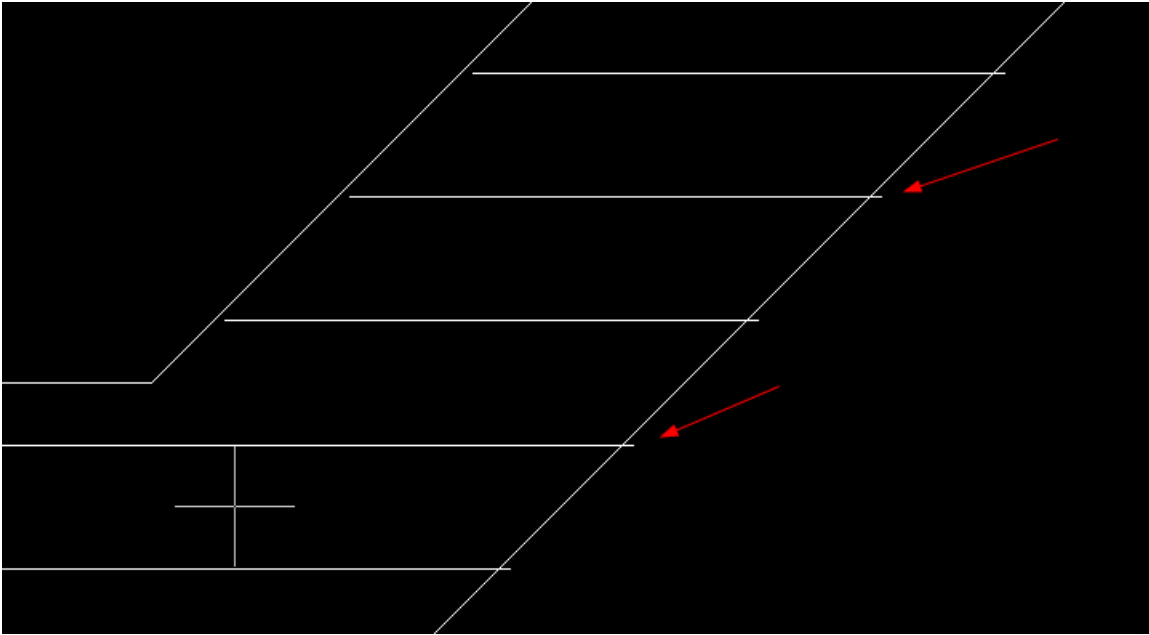
- **Netlist Extract Improvement**

The contour netlist extract has been improved to better detect midpoints if several pads are overlapping and better handling of obround pads.

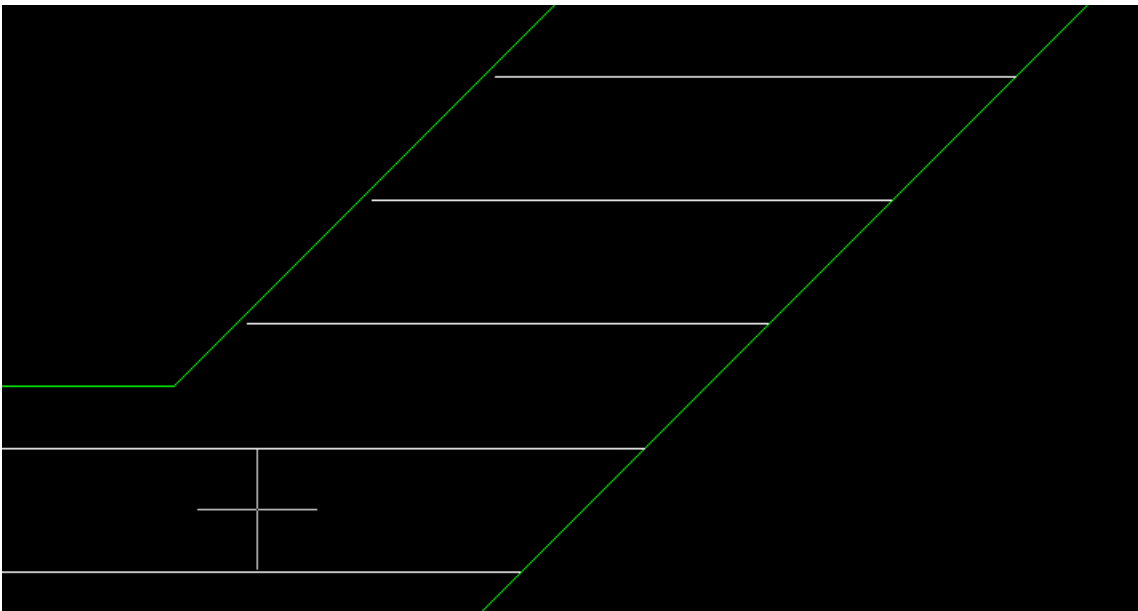
11.0.74:

- **Improvement in Trim Selected Traces**

The "Clip" operation of "Edit/Trim Selected Traces" has been improved. It now clips traces extending by small amounts over another trace even at points that are not close to the endpoints of the other trace. Previously the clipping was done only for situations where traces intersected close to their endpoints. This new method is useful to clean up inaccurate polygon fills, such as the one shown in the image that follows:



The above image is shown at zero-width to expose the nature of the fill. After the trim operation, the result will look as shown in the next image:



- **Automatic Lead-In Generation**

A new command has been added for automatic lead-in/out generation for routing paths in the selection that are closed, do not already have lead-in and have compensation set. The compensation setting is used to determine the side of the path where the lead-in is generated. The command is "Tools/Routing/Auto Lead-In". There is one parameter for this operation that can be set by the user, the offset (height) off the path where the lead-in starts ("Tools/Routing/Lead-In Options/Auto Lead Offset From Edge").

- **Proximity Sort For Routing Paths**

A new command has been added to sort the routing paths by proximity so as to shorten the total distance

traveled by the routing tool ("Tools/Routing/Proximity Sort"). This command will change the precedence numbers of the routing paths. This sort is also done (automatically) when a rout file is exported, but in this case only paths with the same precedence number are sorted. Here is a summary of the export order: ascending tool numbers (smaller tool number first), ascending layer number, ascending precedence number (lowest precedence numbers first, except for precedence 0 which means "don't care" and which is exported last) and finally (within the same precedence) the order is by proximity.

11.0.71:

- **Tool For Panelizing**

A new command has been added: "Edit/Layers Stackup/Combine By Board Number". This command operates on all visible layers and, if several layers have the same board layer number, it will combine those layers into one (merging into the first layer with that board layer number). Although this is a general purpose command that can be used for other purposes also, its main use is to facilitate panelizing two (or more) different jobs.

The jobs are usually on separate sets of layers but one wants to combine them into a set of panel layers. For this one can apply the following steps:

1. Place the second set of layers at the desired X and Y location by using one of the move commands..
2. Set the board layer numbers for each layer in the second set so it matches the board layer numbers in the first set. This can be done in the "Setup/Layers" dialog.
3. Execute the "Combine By Board Number" command.

- **New Global Selection Filter**

A new filter has been added for selecting only paint elements (ignoring scratch elements). It is called "Paint Elements". Previously there was only a filter for selecting "Scratch Elements".

- **Improved Routing Path Compensation**

Compensation computation has been improved for paths containing slots (where a path reverts onto itself).

11.0.59:

- **Feedback during Move and Copy**

For operations that move or copy the selection the cursor will drag a box representing the limits of the selection (to provide feedback to the user). These operations are: "Edit/Edit Selection/Move", "Edit/Edit Selection/Copy" as well as "Edit/Copy" and "Edit/Cut".

- **Shaving Pads**

A new command has been added which activates a tool for shaving pads ("Edit/Shave Pads"). It is a two step operation: first click on the pad to shave, then click at the edge of the desired shave (inside the pad). The width of the shave can also be typed while in the second step by pressing "X". This tool is useful for correcting occasional DRC/DFM errors.

- **Including the Frame in Printed Output**

An option has been added to the printing dialog to include the frame in the printed image. It is "Print Frame also".

11.0.47:

- **Export of Excellon Format 1**

An option has been added to allow selection of Format 1 or Format 2 export of Excellon files. Previously data was exported in Format 2 only.

- **G45-G48 Sieb & Meyer Export**

When exporting rout data, routing paths that are full circles will be exported using the G45 through G48 commands as follows:

- ⇒ G45 if the path is CCW and compensated Left
- ⇒ G46 if the path is CW and compensated Right
- ⇒ G47 if the path is CCW and compensated Right
- ⇒ G48 if the path is CW and compensated Left

These commands are designed for exporting Holes (G45 and G46) and Cards (G47 and G48).

- **Netlist Extract and M Pads**

When extracting a contour netlist, custom pads (M shape) are converted to C shape pads before the extract is computed. This assures better results for midpoint computation and also generates better output files for the ETest products. The conversion is the same as can be done manually using "Edit/Edit Selection/Special/Convert M-type Pads to C-type".

11.0.42:

- **Large Offsets in Gerber and Drill Files**

There are files where all the data is offset by a very large amount which violates the CAM data limits of 1000 inches (or 10m in metric). Now such data can be imported. One needs to use "File/Import/Guess" ("F4") or "File/Import/Directory" for this purpose.

11.0.28:

- **Easy Way to Align Reference Layers for NetCompare**

NetCompare works by comparing two sets of layers: the reference (REF) and the data to be tested (NUT). Occasionally the REF layers are offset from the NUT layers. There is a new command which aligns the REF layers when they are offset ("Edit/Move Layers XY Position/Auto-Align REF Layers"). This command will try to match the position of holes in the drill layer of the REF set with the drill layer of the NUT set (each set must contain a drill layer). If it finds a matching offset for the drill layer, the REF drill layer and all the other layers in the REF set are moved by that amount. This usually results in the data being properly aligned.

11.0.22:

- **Printing in Screen Colors**

A new method has been added to the "File/Print" dialog: "Screen colors". This will print the image the same way it is displayed on the screen. Since the background of the image is black, this will waste a lot of ink when printing to paper, but it is useful when printing to files, such as PDF files.

- **Display of Rout Path Direction**

One can now visualize the the direction or orientation of routing paths. Arrows will be overlaid over the path to show the direction of the routing tool. This can be turned on or off using "Tools/Routing/Show Direction and Plunge". There is also a button for this in the routing toolbar. This command replaces the existing "Tools/Routing/Display X's at Plunge Points". X markers are still being displayed at plunge

points when this command is active, in addition to the arrows showing the direction.

11.0.14:

- **Improved Computation of Compensated Rout Paths**

The computation of compensated routing paths has been improved.

11.0.9:

- **Quick Way to Insert Round Polygons**

A new tool is available to insert round polygons. It is "Insert/Circle/Polygon" and there is also a button for it on the "Insert" toolbar (shown as a filled circle). The operation takes two clicks: one for the center of the circle, the other for a point on the circumference.

When the goal of the operation is to insert round pads, the "Insert/Pads" tool is preferable, but the insert round polygon can be useful when making corrections (filling in areas or scratching away unwanted copper).

11.0.1:

- **New Tool for Selecting a Set of Connected Elements**

"Select/Connected" is a new tool that allows selecting elements that are connected (touch or overlap).

Once the tool is activated one can click close to an element and that element and all elements in that layer which are connected to it (either directly or indirectly via other connecting elements) are selected. This command works only on visible elements and ignores scratch elements.

Possible uses for this command:

- ⇒ To generate a board outline. One can click next to one element in the board outline and all connected elements will be selected. This is most useful when traces in the outline do not line up perfectly on top of each other (they intersect or overlap). If one wants to generate a "correct" outline after such a selection is made, the "Edit/Trim Selected Traces" command can be used.
- ⇒ To check netlist connectivity. This is not really a netlist, because it only works on one layer and ignores scratch elements, but it can be helpful in the case of simple designs.

- **Automatic Alignment of Drill Layers**

A new command ("Edit/Move Layers XY Position/Auto-Align Drill Layer") aligns the drill layer (of type DRI or PTH) to a component layer (CPU or CPL). It will attempt the alignment by trying to match all plated holes in the drill layer to pads in the component layer. Each hole needs to be matched for this operation to succeed, so the component layer needs to have at least as many pads as the drill layer. Only visible elements are considered.

- **Improved Setup of Via Layers for Netlist**

The setup of VIA type layers for Contour Netlist extract has been improved. A via layer needs only be present once in the stackup and one can specify which layers it crosses. Previously copies of the layers had to be made and placed in between any two circuit layers it crossed. This can save a lot of time when setting up jobs for netlist extract. The old method can still be used. There is an application note describing this in more detail at: [ftp://ftp.pentalogix.com/Documents/AppNotes/Netlist/Via Layers and Netlist.pdf](ftp://ftp.pentalogix.com/Documents/AppNotes/Netlist/Via%20Layers%20and%20Netlist.pdf).

- **Netlist Export of Obrounds**

A new option has been added to control how obround pads are exported during contour netlist extract and export. These pads are exported as rectangles, but now the user has the option to pick the inscribed (smaller) or external (larger) rectangle.

10.6.71:

- **Improved DFM**

A new command has been added to the “DFM” menu: “DFM/All Violations”. This allows for spacing and width violations to be run at the same time and also has a new set of rules for drill related violations (angular ring and clearance).

- **A Quick Way To Insert A Rectangular Polygon**

“Insert/Rectangle/Polygon” will insert a rectangular polygon. The user can pick the two corners of the rectangle. This can be used as a quick way to make corrections to a design by filling in (inserting a paint polygon) or deleting (inserting a scratch polygon).

- **Order of 90-degree Trace Pair**

When inserting 90-degree trace pairs (“Insert/Special Linear Trace/90 Degree Trace Pair”) one can toggle the orientation of the first trace with the Shift key. Previously the first trace was always horizontal.

- **Dimensioning Improvement**

When inserting orthogonal (horizontal or vertical) dimensions (“Insert/Dimension”), the choice between horizontal and vertical is now made from the location of the third picked point (for the text position). Previously this was determined from the endpoints of the measurement (the first two picked points).

- **Export Warning for Small Apertures**

A warning will be generated when exporting Gerber 274X files with apertures smaller than a user specified value. This option can be turned on or off in the export “Options” dialog. It replaces the warning for exporting zero-width apertures (one can still get it by setting the “smaller than” aperture dimension to 0).

10.6.50:

- **Layer Names Shown in Layers Toolbar**

The Layers Toolbar (“View/Toolbars/Layers”) now shows the layer names also in addition to the layer number and color. The name appears to right of the color box, so one needs to widen the toolbar to show as much of the layer names part as desired.

One can widen the layers toolbar by dragging the right edge with the mouse. Once set to a new width that width will be remembered. It is a user preference as to how much screen space one wants to give up to display layer names.

- **Improved Width Violations Detection and Automatic Correction of Slivers**

“DFM/Feature Size Violations” operation has been improved. It is now faster and reports more violations (previous algorithm reported only one violation per copper area).

A new feature has been added as well: if slivers are not being ignored, then one has the option to correct them during the review process by pressing “Correct” or “Correct All”. Slivers often appear in negative plane layers. A sliver is a small piece of copper for which the maximum dimension is smaller than a user specified value.

- **Inserting Arc Chamfers of Given Radius**

A new command has been added to allow inserting arc chamfers of a fixed radius, specified by the user: “Insert/Chamfer of Radius”. The chamfer is added at the joining point of two linear traces. After the tool is activated, the user needs to click close to this joining point and an arc chamfer is inserted.

- **Showing Scratch Elements in Contrasting Color**

There is a new command to set the display mode so that it shows scratch elements as paint in a contrasting color. This is sometimes useful when trying to understand the paint-scratch setup of a layer. Note that this works only on combined (mixed-polarity) layers and not on whole layers that are marked as scratch.

The shortcut to toggle the mode is “Ctrl+H” and the command can also be found in the “Setup/Visibility and Display” as “Scratch contrast”.

- **Intersection of Two Layers**

A new command to compute and generate the polygons that result from intersecting two layers. The intersection is defined as the areas where both layers have “copper”. One of the layers (the current layer) is replaced with the polygons that result from intersecting it with another layer (the user is prompted for the other layer).

10.6.34:

- **Tool for Inspecting Net Midpoints / Endpoints**

The tool is “Tools/Netlist/Inspect Midpoint Stackup”. After it is activated, one can click at a pad location and the status line will show the midpoint/endpoint state of the pads at that location in the layers CPU, CPL and DRI or PAD. The netlist information must have been previously imported or extracted in CAMMaster (with the identify midpoints option checked).

- **Marking a Midpoint**

A new tool to insert a marker at the midpoint between two points picked by the user. These two points are usually picked by aligning to elements, but the tool is general purpose. It is activated with “Go/Mark Midpoint”. It stays active until canceled. Useful for creating reference points when constructing complex geometries.

10.6.30:

- **Comparing Groups of Layers**

The command “Tools/Compare Layers/Find Differences” can now compare two groups of layers (layer by layer). Each layer in the first group is compared against a corresponding layer in the second group. This makes it easier to compare different versions of the same job. If you need to compare two jobs (or two sets of layers) and want the dialog to be set up automatically with the proper ranges, load the first job then leave one or more blank layers and finally load (import) the second job after the blank layer(s). The layer compare dialog will be set up so that the two ranges to compare will show correctly.

One can still compare just two layers at a time, like in the past. One can also manually enter the layers to compare into the dialog boxes. Note that only visible layers will be compared.

- **Determining Board Area**

The Routing Path Properties dialogs now have a new item called “Enclosed Area”. This is the size of the area of the internal region of a closed routing path (the area that is cut out by the path). For the outermost path, this is the area of the board.

Tools that show this dialog are “Tools/Routing/Selection Properties” and “Tools/Routing/Properties Tool”. If the data applies to more than one path, then the sum of all enclosed areas is shown.

- **Improved Break Tabs**

Two new features have been added to the routing break tabs setup dialog (“Tools/Routing/Insert Break Tabs”).

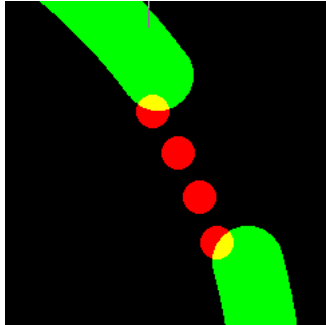
- 1) “Distributed by Distance” as a method for adding multiple tabs in one step: this will compute the

number of tabs that fit equally spaced given the user specified “Distance between Tabs” and then generate that many tabs.

2) New way of generating perforations (mouse bites):

- a) The holes are now offset from one of the edges. They used to be situated on the center-line of the path. The offset distance can be specified by the user and the side is picked by clicking close to one of the two sides of the routing path. Negative offsets move away from the center-line of the path, positive offsets towards the center-line.
- b) The number of holes that result can be specified one of two ways:
 - by entering it directly as the desired number of holes in the dialog or
 - by entering the desired gap which will result in the number of holes being computed from the geometry of the setup.

Geometries like the one shown below can be easily generated:



- **Net Midpoint Info**

The command “View/Info/Elements Overlapping Cursor” (also available in the context menu – mouse right click over the view) will now also show the midpoint/endpoint state of a pad (if net info is present in the job). This facilitates checking the midpoint/endpoint state in a layer stackup at a given point.

- **Faster Netlist Midpoint Computation**

Contour netlist midpoint computation is now significantly faster, especially for jobs with many inner layers.

10.6.0:

- **New Method for Correcting Self-Intersecting Polygons**

When the input data contains self-intersecting polygons, they need to be corrected so that they can be used in later steps of the CAM process. This correction can happen when the data is imported (the user is prompted for doing the correction during the import) or can be done later on selected polygons (with “Edit/Edit Selection/Polygons/Make Well Formed”). This existed already in previous versions of the software. This version adds a new correction method. There are now two methods available, which can be picked by the user via the command “Setup/Self-intersecting Polygon Correction”. These two methods are called:

- ⇒ Even-odd method
- ⇒ Non-zero fill method

It is easiest to understand what they mean by looking at the images in the dialog used to pick the method. The most common method (and the only one previously available) is the “even-odd method”. However, some software packages generate polygons that need to be corrected using the “non-zero fill method”.

- **New Command to Select Electrically Connected Elements**

This command is a selection tool. Once active, one can click on an element and all elements that are electrically connected to it, through all layers, are selected. The command to activate the tool is “Tools/Netlist/Select Electrically Connected Elements”. When first activated, it will extract the netlist information for the currently loaded job. All conductive layers will be considered, whether visible or not at the time the tool is activated. This means layers of type SIG, CPU, CPL, all power and ground layers, DRI, PAD, PTH, VIA and VID. If you have other layers that should not be considered in the computation, you should make sure that they have a different type. If they are “work layers”, map them to OTH or NFL.

Once the netlist info is extracted, the tool can be used repeatedly until modifications are made to the database, when the netlist will again be extracted (automatically) to match the changes made.

To make it easier to select elements, board layers that are negative or of mixed polarity (paint and scratch) will be converted to polygons while this tool is active. They will be restored to their previous state after the tool is deactivated.

An application note describing this command in more detail is at: <ftp://ftp.pentalogix.com/Documents/AppNotes/Select Connected Copper.pdf>.

- **Polarity Combine / Separate Improvements**

The operation of the commands which separate layers of mixed polarity or combine layers (“Edit/Paint and Scratch”) has been improved. The layer stackup of layers which are not involved in the operation is preserved. Previously, empty layers were eliminated during “Combine All”.

Undo has been added to all the operations in this group.

10.4.31

- **New Tool Type**

For NC tools (D Codes of shape “N” and “Setup/Tool Codes”). The tool type is “**Rout**”. It is intended to be used for routing paths, to differentiate from other tool types. The other types are: “Plated”, “Non-plated”, “Tooling”, “Via” and “?”. Note that .bin files containing such tool types cannot be opened by older versions of the software.

- **Deleting Holes from D Code Shapes**

A new command has been added to the “Setup/D Codes” (F5) dialog. It is “Operations/Remove Holes”. It will remove holes from all D Codes which have holes. This includes standard shapes with holes, like C (an annular ring), and custom shapes also (M). The operation works on the D Code (or D Codes) selected in the dialog (shown in blue background).

- **Drill Path Sorting and Viewing**

New commands have been added for viewing the path (order of holes) of drill holes in a drill layer and for sorting the holes.

“Tools/Drill/Show Path” will display the path, given the current order of the holes. A dialog is displayed that allows navigating the path.

“Tools/Drill/Proximity Sort” and “Tools/Drill/Serpentine Sort” will sort the holes according to two sorting methods.

⇒ “Proximity Sort” will start from the lower left corner of each tool area and then continue to the next closest hole until all the holes of that tool are visited.

⇒ “Serpentine Sort” will sort in a serpentine fashion (in bands or strips).

Usually “Proximity Sort” will generate a better result, but this is not guaranteed.

Notes on drill holes order: sorting is best done just before exporting the drill files, because the order can be disturbed by later editing operations and even by selecting and deselecting elements.

To specify first and last holes for a tool number the recommended method is to separate the drill holes

into adjacent layers and then export with the “Combined” method.

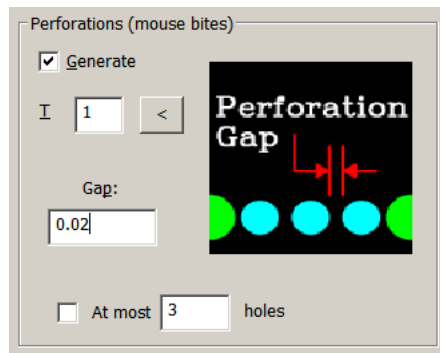
For instance, if one has holes for tools T1 and T2 in layers 1, 2 and 3, when exporting the three layers together as “Combined” the order would be:

holes of tool number T1 from layer 1, then T1 from layer 2, then T1 from layer 3, followed by T2 from layer 1, then T2 from layer 2 and finally T2 from layer 3. This way has the additional advantage that one can easily see the order of holes via the layer colors.

If there are routing paths in the layers exported they will be exported after the drill data. The only exception is when exporting “Combined”. Then one can have routing paths in layers preceding any layers with drill data and these will be output first.

- **Perforations for Routing Path Break Tabs**

“Tools/Routing/Insert Break Tabs” has new options for inserting perforations (also commonly known as mouse bites). The parameters currently available are shown below:



- **More Components Information**

A new command has been added to display component related text, when such information is available. It is “View/Info/Components Text”. Currently this information is extracted when importing ODB++ files (if they contain components information).

10.4.30

- **Dragging the view**

The view can be dragged by holding the mouse left button down and dragging. This has been available for some time in our software but required holding down the Ctrl and Alt keys while dragging. We have now changed the code so one can drag without Ctrl and Alt being pressed. This will work only if no command-tool is active, such as Select or others. If a command-tool is active then pressing the left mouse button will activate the command and to drag one needs Ctrl and Alt pressed. Note also that if you want the view to be dragged in real time, you need to enable it with “Setup/Preferences/Draw View In Real Time”, otherwise the view will only be updated at the end of the drag. When there is a lot of data in the current job dragging in real time may cause flickering because it takes longer to redraw each intermediate step.

- **Export Rout and Drill combined**

The “File/Export/Rout” commands have been removed. Rout files can now be exported with “File/Export/Drill & Rout”. Any layers that have rout paths and are exported with this command will generate rout data in the output. This allows combining drill and rout data into the same file.

When files are exported with the “Layer by layer” method, each layer generates a separate output file in which drill holes are exported first followed by routing paths.

When files are exported with the “Combined” method, the data in all the layers is output into one file.

The following rules apply to the order of the output for the selected layers:

If some starting layers in the group of layers to output have only routing paths, then those paths are output first.

For all the following layers drill holes are exported first. They are ordered first by tool number and then (within the same tool number) by layer numbers. This is an easy way to separate (for a given tool) holes that one wants to drill first and last.

At the end come all the routing paths found in all the layers other than the ones from step (1). They are ordered by their precedence number. Paths which have the same precedence will be further ordered by their tool number and for the same tool number, by the layer number.

- **Improved ODB import**

More support for new features in ODB++ V7.0.

10.4.0:

- **Copy, Cut and Paste**

The standard Windows commands for copy, cut and paste have been implemented, with the usual shortcut keys: Ctrl+C, Ctrl+X and Ctrl+V. They are in the “Edit” menu and have buttons in the main toolbar.

Copy and cut work on the selection, so one first needs to select elements. They both copy the selection to the Windows clipboard and cut also deletes the selection (similar to “Edit/Edit Selection/Delete”). Later on, the clipboard data can be pasted (into the same program or another instance of this program or another PentaLogix CAM package). The data will be pasted into the same layers where it was originally, but the location (X, Y) can be different. The rule is that the position of the cursor over the selection when the copy was done (copy or cut) will match the position of the cursor when the paste is done. So a cut and paste can be used to move selected data from one (X, Y) location to another: select, position cursor, cut, move cursor, paste.

- **Pasting CAM Images Into Documents**

The current view can be copied to the Windows clipboard using “View/Copy View To Clipboard”. The resulting image can then be pasted in a variety of documents, such as a Word file, an email etc. without needing a screen capture program.

- **Button Names for “Favorite Commands” Toolbar**

Buttons in the Favorite Commands toolbar can now be given names in addition to mapping them to bitmap images. After right-clicking on the button, the new option is called “Change Text”. Another new feature of the Favorite Commands toolbar is that the first nine buttons can be activated with shortcuts: Alt+1 through Alt+9. And finally, the button sizes have been increased from 16 by 16 pixels to 24 by 24 pixels to make it easier to generate bitmap images for the buttons.

- **Alphanumeric Net Names**

An option has been added to export alphanumeric net names (instead of net numbers) in IPC files (“Tools/Netlist/Extract Contour”). The option is in the “Export” tab and is called “Net numbers output as N<number>”. This will cause, for instance, net “223” to be output as “N223”. There are apparently certain software packages which do not like net names that contain only digits.

10.2.71:

- **Toolbar for Favorite Commands**

“View/Toolbars/Favorite Commands” displays a customizable toolbar for favorite commands. This is useful for often used commands which are nested deep in the menu structure and can be accessed quicker this way. The toolbar is described in more detail in the document: “Favorite Commands Toolbar.pdf”. Look for it in the installation folder or on our FTP site (<ftp.pentalogix.com/Documents/AppNotes>).

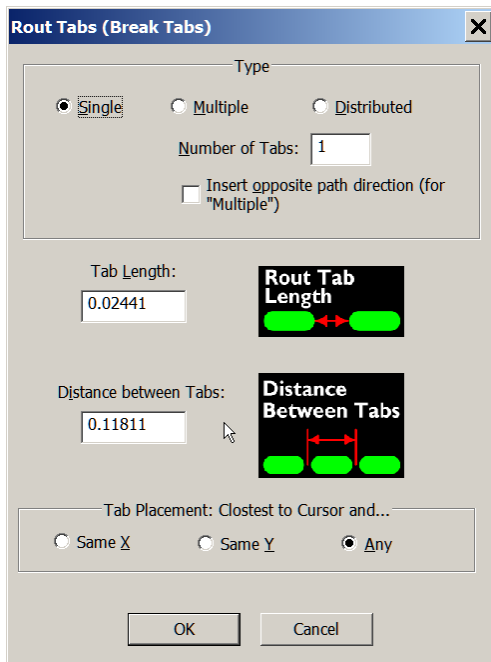
10.2.45:

- **Routing Toolbar**

A new toolbar has been added with many commands for creating and editing routing paths. "View/Toolbars/Routing".

- **Routing Tabs (Break Tabs)**

A new command to insert break tabs into a routing path ("Tools/Routing/Insert Break Tabs"). It activates a tool that can be used to insert one or many break tabs at the spot where the user clicks on a routing path. The parameters for this operation are shown below:



- **Short Edges in Routing Paths**

The routing path property dialogs for the "Tools/Routing/Selection Properties" and "Tools/Routing/Properties Tool" have a new item called "Shortest Edge". This will display the length of the shortest edge in a path or a collection of paths. Short edges (straight or arc) can cause problems with certain routing machines.

- **Easy Way to Navigate Selected Elements**

A new command ("Select/Navigate Selection") has been added to allow for easy navigation of the currently selected elements. It brings up a dialog with the usual "Next" and "Previous" buttons that allow for easily stepping over all selected elements.

Note that if one changes the selection while the navigation dialog is shown, the new selection will not be reflected in the dialog (one needs to close and reopen the dialog).

- **Finding Intersecting Traces**

A new command to select traces that intersect at points which are not their endpoints (traces which cross). The command is "Select/Intersecting Traces". Only active elements will be considered, so the "Only" check-marks etc. will be taken into account. This command is useful as a DRC check.

- **Shortcut for Selecting Crossing Elements**

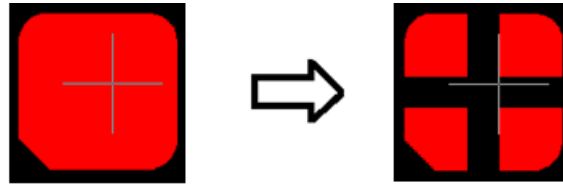
A new shortcut has been added for toggling the selection of elements which cross the frame. This shortcut

is "Ctrl+Shift+S". The button is in the Selection toolbar.

- **Stencil Improvements**

The stencil module ("Tools/Stencil Apertures") now works on all pad shapes, not only rectangles (except for the homebase, which can only be applied to pairs of rectangular pads).

An example for "Windows" is shown below:



- **Export NC Rout Files**

- ⇒ A new option has been added to output F commands. The F command defines the worktable feed rate (or routing speed).
- ⇒ For export Excellon a new option has been added to turn on or off the output of G90/G91 (absolute/incremental coordinates) and M71/M72 (metric/English) commands. Some routing machines have trouble accepting these commands.

10.2.18:

- **"Edit/Edit Selection/Polygons/F To Polygons"**

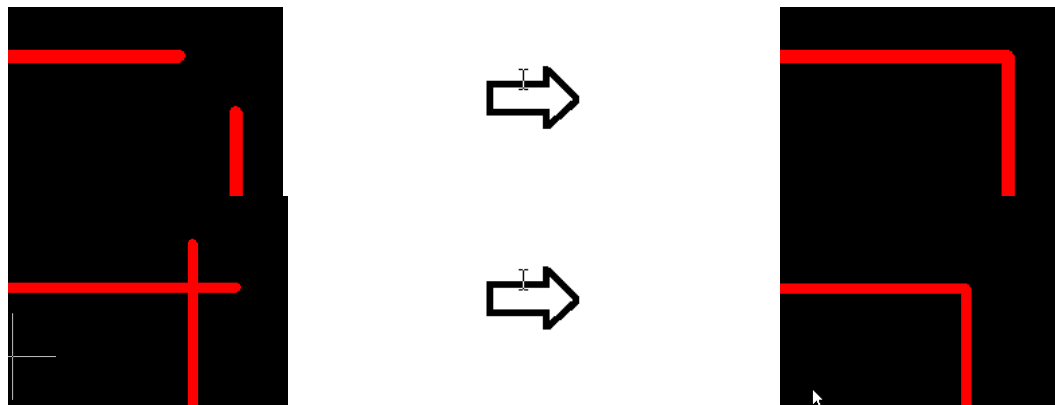
A new user option has been added "Path can continue at overlap". If checked, will enable special processing of overlapping traces, so that a contour can continue at vertices where traces overlap. If not checked, will force contours to use the traces as they are.

10.2.0:

- **New Commands For Trimming Traces**

These are commands for correcting pairs of traces that either are close to each other at endpoints (but do not connect) or overlap slightly at endpoints. These situations sometimes arise because of rounding errors in the CAD output or inexact manual insertion of traces.

The two types of problems ("Connect" and "Clip") and their corrections are illustrated below:



These problems can be corrected in one of the following ways:
by picking the two traces (with a mouse click or the Enter key). This is "Edit/Trim Traces" and acts as a tool (it stays active until the user cancels it).

by first selecting all the traces that one wants to correct and then issuing the command “Edit/Trim Selected Traces”. A dialog comes up to allow the user to specify which operation is to be done: “Connect”, “Clip” or both and a parameter which determines how far apart the endpoints can be for connecting. There is a button for the tool version of the command in the “Edit” toolbar. Both linear and arc traces can be trimmed.

- **Converting Negative Layers To Positive**

Negative layers can now be converted to positive by using paint and scratch. A paint polygon (or several paint polygons) will be inserted before all data to represent the background copper and then the polarity of elements in the original negative layer will be changed so that the result looks the same.

If the data on the negative layer contains the board outline, it will be used to generate the background copper polygons. Otherwise the frame will be used for this purpose, so it needs to be positioned before the operation to surround all data on the layer.

This operation is useful if one later needs to combine negative and positive data in a panel.

The new commands are:

“Edit/Convert Negative To Positive”: allows the user to specify which layers to convert

Right click on layer in the Layers Toolbar, then “Convert To Positive”: converts the picked layer only

- **“Insert/Into Layers”**

A new command to allow users to specify the layers where new elements will be inserted using the “Insert” commands. The user can pick one of two modes:

⇒ Active Layers

This is the same as insert used to work previously. Elements will be inserted in the active layer or active layers depending on the state of the “Only” check-mark for the current layer (in the Current Element toolbar).

⇒ Specified Layers

User can specify the layer (or layers) where elements will be inserted. Changing the current layer or the “Only” check-mark does not affect where the elements are inserted. This new mode allows, for instance, to insert into one layer while aligning to elements in another layer.

In both modes, the layers that will receive elements will be marked with a “>” before the layer number in the layers toolbar. Since more room is needed now for displaying the layer number text, the color box will be narrower, but one can adjust that by widening the layers toolbar (drag the right edge of the toolbar).

- **“DFM”**

New DFM commands have been added. They are in a top-level menu called “DFM”.

The new commands are for spacing and feature width (copper width) computation. There are commands to determine minimum spacing and width and commands to find spacing and width violations.

All of the commands allow the user to pick the layers which will be inspected and also whether the inspection is to be done only inside the frame or in the whole layer. Inspection is done only on visible elements, so one can further limit the scope of the operation by making certain elements invisible (for instance one can make pads invisible prior to starting one of these DFM operations).

The commands are:

“Find Minimum Feature Size”

Will find the minimum width (feature size) of copper on a layer. Only one minimum width will be reported per layer, so, if it appears in several spots on a layer, an arbitrary one will be picked.

“Find Minimum Spacing”

Similar to “Find Minimum Feature Size”, but will find the smallest spacing between two different copper areas in one layer. Spacing between parts of a connected area of copper is not taken into consideration.

“Feature Size Violations”

Checks for feature size (width) violations: locations where the copper width is narrower than a user specified value (“Feature Size”). There is an optional parameter for dealing with slivers (“Copper Sliver Size”). Slivers are narrow copper entities: areas of copper whose width does not exceed “Copper Sliver Size” at any point. If one does not want to deal with slivers as a special case, one can set “Copper Sliver Size” to 0.

There are three modes for this DFM check:

“Check for feature size violations and ignore slivers”

This will silently ignore slivers and check for width violations only on the remaining (non-sliver) areas.

“Detect slivers only”

This will only look for slivers and report where they are. Corrections are provided, which erase the slivers.

“Check for both”

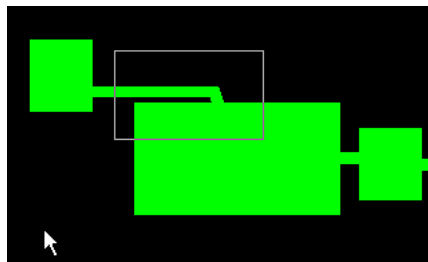
Slivers are detected and corrected (as in the second method) and width violations are reported on the other copper areas (as in the first method).

“Spacing Violations”

Checks for spacing violations: locations where the space between two distinct copper areas or edges of one copper area is smaller than a user specified value.

Violations of the second kind (same copper area) are called acid traps or resist slivers. They are further separated into two kinds, open and closed, each with a separate spacing violations distance. Examples of the two kinds are shown below:

Open

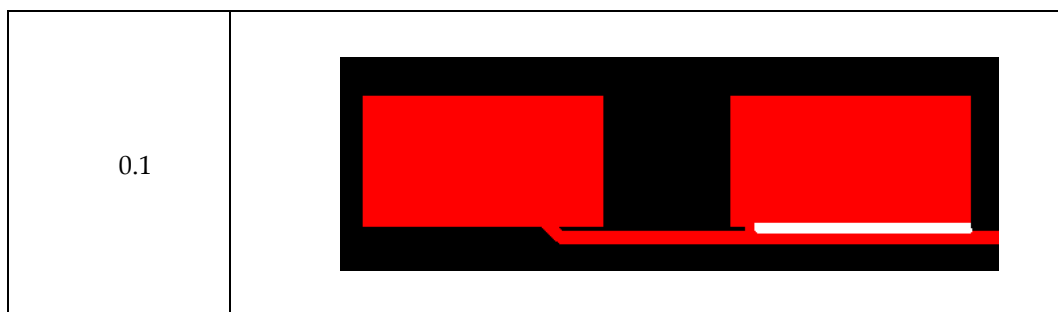


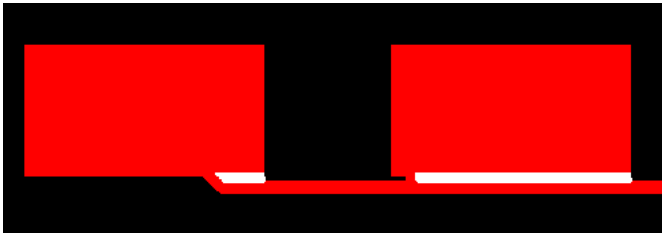

Closed



The algorithm will generate a correction for the acid trap which will fill in the opening.

For open acid traps the user can specify an additional parameter: an aspect ratio. It is a number between 0 and 1. Making the number smaller (towards 0) will limit the detection to more elongated acid traps. Making the number larger (towards 1) will allow less elongated violations to be detected as well. Below are examples of corrections (white areas) with various values for the aspect ratio constant:



0.35	
0.8	

- **“File/Open Options”**

Groups options that apply to “File/Open” (reading in .bin jobfiles).

“Ignore Preferences From Jobfiles” existed previously but was located in “Setup/Preferences”.

“Do Not Restore Custom Colors” is a new one and will cause the software to ignore custom color definitions (RGB) from jobfiles. This helps users who want to always preserve their own custom color setups (done with “Setup/Layer/Colors” -> “Redefine Color”).

- **New Option For Selection**

A new user-settable option has been added to enable the selection using a frame of elements which cross (intersect) the selection frame. When this option is off, only elements which are completely inside the frame are selected (this was the previous behavior of the software).

The option can be found in “Select/Options” and is called “Select also elements that cross the frame”.

There is also a button for this in the Selection Toolbar (“View/Toolbars/Selection”). It has been added next to the “Inside” button.

- **Routing Paths Length**

In “Tools/Routing”: “Selection Properties” and “Properties Tool” will display the length of the routing paths in addition to the other information. This makes it easy, for instance, to determine the total length of all routing paths.

- **“Tools/Netlist/Find Path Connecting Two Holes”**

A new command to trace the path (through copper) connecting two holes from the drill layer (or padmaster layer). Copper connectivity through all visible board layers is considered. At the end, a dialog appears with text that shows how the path goes from the first point to the second point.

Before activating this command the layers need to be set up as if for a contour netlist extraction, with all board layers that are to be considered made visible. Also two holes need to be selected in the drill layer (DRI) or padmaster layer (PAD) and the selection must contain only these two pads. The command will compute the connecting path between these two points.

The current implementation does not yet take into account via layers, but that will be implemented in the near future.

This command is very useful for finding where two nets short. Visual inspection is sometimes difficult when the net crosses many board layers.

When the command is first executed, the algorithm computes copper connectivity for the whole data. This can take some time. Once computed, this info is stored and does not need to be recomputed for subsequent invocations of this command (unless the job data is modified). The polygons used in the computation can be visually inspected by using the command "Tools/Netlist/Import Copper Polygons" (this command is also available after a contour netlist extract).

- **"File/Import" ("Guess", "Directory" and "Zip")**
 - ⇒ the interpretation of the format of Gerber files has been improved.
 - ⇒ for Sieb & Meyer drill files: the format guessed (or set by the user after the guess) will override the format implied in the Sieb & Meyer file (for instance format %%3000 automatically implies metric, trialing zeros, 3.3 and this can now be overridden).
- **"Setup/Frame/Surround All Data" (Ctrl+W)**

State of "zero-width" display ("O") is taken into account when computing the size of the frame.
- **"Tools/Stencil Apertures"**

The shape computation has been improved.
- **"Tools/Snap To Padmaster"**

Traces could be deleted when both endpoints were snapped to the same point. The algorithm has now been changed so in cases like this only one of the endpoints will be snapped.
- **"File/Export/Rout"**

Compensation commands (G40, G41 or G42) are now output before each path. They were previously output only when the compensation kind changed, but the Excellon G00 command turns compensation off, so that compensation needs to be re-output after each G00.

10.0.30:

- **"Tools/Netlist/Extract Contour"**

An option has been added to allow the user to pick how the midpoint state of drill holes is computed. This only applies to drill holes which have covering pads on both the CPU and CPL sides. First the midpoint state of the CPU and CPL pads is computed and then the midpoint state of the drill hole is determined based on the user picked choice:

 - ⇒ "Drill hole is mid when BOTH are mid": the drill hole will be marked as a midpoint only when both the CPU and CPL pads are midpoints.
 - ⇒ "Drill hole is mid when ANY is mid": the drill hole will be marked as a midpoint when either the CPU or the CPL pads are midpoints.

10.0.26:

- **"Tools/Convert Drawn Pads/Global"**

The algorithm has been improved.

10.0.10:

- **Shortcut keys for Layers Toolbar**

An option has been added to enable shortcut keys for the layers toolbar. This is in the layers toolbar context menu and is called "Shortcuts enabled". Shortcuts were removed a few versions back because they were deemed dangerous, as they have a different meaning than the same shortcuts in the main view. However, some users complained and wanted them back so now they are restored (if enabled by the

user).

The shortcut keys available are the same as in the past: “V”, “N” and “S”. Note that “V” and “N” work on the whole board layer, just as in the past.

10.0.1:

- **“Edit/Move Layers XY Position/Move Layer Tool”**

A new tool has been added to facilitate layer alignment. The command will activate a tool which consists of two steps:

First the user picks an element in the layer that he desires to move. This is done by moving the crosshair close to the element and then clicking (or pressing Enter), similar to how one selects an element.

Next the user picks the corresponding element in another layer the same way as in the previous step.

At this point the layer of the first element will be moved so that the first element is now on top of the second element (and hopefully the layers are aligned). The whole cycle can then be repeated again for moving another layer. When done, press Escape to cancel the move layer tool, or deactivate it by clicking on its menu entry. This command has the shortcut key combination Ctrl+Shift+Q.

- **Explicit Units**

All the dialog items that refer to a coordinate or a dimension now accept numbers followed by optional units, independent of the current units in use. For instance in the “Move Cursor To” dialog box one could enter “2.3cm” for X, even though the current units may be “inch”. This would cause the cursor to move to 0.905512 inches. The acceptable values for the units are the same as the ones shown in the “Coordinate Units” box: “inch”, “mil”, “cm” or “mm” or any prefix of them which is unambiguous (for instance “in” or even “i” will work instead of “inch”).

- **“Setup/Preferences/User Confirmation For”**

Confirmation for dangerous operations. A new group of preferences that can be turned on or off. When on, they will cause the corresponding operations to prompt for user confirmation before they are done. Currently two such operations covered:

⇒ **Delete Selected:** will prompt for “Edit/Edit Selection/Delete”

⇒ **Drag And Drop Layers:** will prompt during a drag and drop in the layers toolbar

These are intended as a safety feature to protect against unintended operations, especially since “delete selected” has a shortcut key (Delete) which can easily be pushed by accident.

More such operations can be added in the future. Please let technical support know which you would like added.

- **“Setup/Preferences/Wheel Forwards Decreases Zoom”**

User preference For those who are used to have the zoom decrease when the mouse wheel is moved forwards and increase when the mouse wheel is moved backwards.

- **“File/Export/Drill” and “File/Export/Rout”, G93-type offsets**

A new option has been added for Excellon export to cause the software to output a G93 (Zero Set) command at the beginning of the file (or the equivalent M49,FA for Sieb and Meyer). It is in the “Options” dialog, under “Output Options” and it is called “Offset”. When turned on, a G93 will be output at the beginning of the file. The value of the offset is taken from the current cursor position, so if you want accurate values, best to disconnect the mouse from the crosshair cursor when activating the dialog. Note that the G93 will be output even if both the X and Y are 0.

- **“Tools/Clip Silkscreen Traces”, new method**

A new method has been added for correcting silkscreen clearance errors. This method inserts scratch polygons into the silkscreen layer to scratch away from interfering silkscreen elements. The old method, which clipped the traces, is still available. The method can be selected by the user in the operation dialog.

- **New Shortcut Key (Ctrl+Shift+U)**
To return to last selection criterion picked previously. This is most often picked from the droplist in the Selection Toolbar ("Any", "Pads", etc.). There are many different criteria in that list and many times one wants to alternate between having one particular criterion picked, say "Polygons" and not having anything picked ("Any"). With this new shortcut one can do this from the keyboard only. After picking a criterion, "U" will pick "Any" and "Ctrl+Shift+U" will bring back the criterion picked before.
- **"Edit/Move Layers XY Position/Move By Displacement"**
This command has been improved to make it easier to align layers. Previously the command dialog included only the X and Y displacements and it moved the "active layers" (as determined by layer visibility and by the "Only" button next to the current layer in the Current Element toolbar). There are two improvements to this command:
 - ⇒ **One can now pick the layers to move in the dialog.** There is a text box where one can type in a comma separated list of layer **ranges**, such as "3, 7-9, 15" (one can also type "All" here). This list is initialized to the currently active layers, so that the command acts as before if the user leaves the layers text box alone.
 - ⇒ **The X and Y displacements** in the dialog are initialized to the result of the most recent distance measurement command (if any). This allows the user to previously do a distance command ("D") to set the move displacement.
- **"Tools/Netlist/Extract Contour", improved midpoint algorithm**
The algorithm for determining midpoints has been redesigned. The new algorithm is both significantly faster and also more accurate in determining midpoints and endpoints.
- **"Setup/D Codes" (F5), custom shapes**
Custom shapes (M-type) setup dialog: now has two added buttons, "Import" and "Export". These will import/export to/from Gerber the current custom shape. These commands were available previously as a separate step in the "Custom Apertures" menu of the main dialog, under "Import From File" and "Export To File". The addition of the new buttons saves some operator steps.
- **"Select/By Intersection", new option**
A new option has been added: "Check only elements at the same polarity level". When on, only elements that are of the same polarity (paint or scratch) are checked against each other and also only when they are at the same level in the paint/scratch sequence. So, for instance, an element in the second paint group of the destination is only checked versus elements in the second paint group of the reference. When one wants all elements checked against all other, this option should be left unchecked.
- **"Edit/Edit Selection/Polygons/F To Polygons", algorithm improved**
Three options have been added **for handling polygon holes (when nesting is requested)**.
 - "Holes cannot overlap": if two holes overlap one of them becomes a container
 - "Overlapping holes are merged": two overlapping holes are merged to make a larger hole
 - "Overlapping holes not merged": the overlapping area of two holes becomes a new container
- **"Edit/Edit Selection/Special/Convert M-type Pad to C-type", improved**
The placement of the resulting C-type pads has been improved. When possible, they will be placed at the center of the M-type pads.
- **"File/Import/Netlist"**
When the "Add feature (if present in 317)" is checked the feature pad will be added to the CPU and CPL layers. Previously they were added to the DRI layer.

- **“File/Import/Guess-Directory-Zip”**
 - ⇒ Now supports the Excellon format for decimal placement (000.000).
 - ⇒ The results dialog has been changed.
 - ⇒ The leading, trailing, all present, decimal present radio buttons will enable/ disable the ‘m’ and ‘n’ labels in a similar way that the Import/Drill & Rout/ options dialog behaves.
 - ⇒ If the size the user selects creates a coordinate value outside the 1000 inch, 1000 cm limit, an error is reported and the “OK” button is disabled.
 - ⇒ Files with no data (drill with tool sizes but no drill data) will now report the board size as “0 x 0” instead of “No Data Found”.

9.8.70:

- **Drag and Drop in Layers Toolbar**

One can now drag and drop layers to reposition them. In the layers toolbar click on a layer and drag it to the desired spot. The insertion mark will be shown before the layer where the drop will occur. The number (and name) of the layer being dragged can be seen in the status line.
- **New shortcut keys (accelerator keys)**
 - ⇒ **Ctrl+Space:** for the main view context menu (or mouse right-click)
 - ⇒ **Ctrl+Shift+F10:** for “View/Visibility/Layers/Only Visible”. This is a command that allows making visible just a group of layers (a comma-separated list of ranges).
- **“File/Import/Gerber” – “Options”**

A new option has been added to the “File Interpretation” tab: “If data follows M02”. It has two possible options: “Make new layer” and “Same layer”.

This applies to the (rare) cases where a file has a Gerber M02 command (End Of Program) followed by more Gerber data. If “Make new layer” is picked, the data that follows is put into a new layer. This is the more common setting (and the behavior of previous versions of the software). Sometimes, when the data that follows belongs to the same layer (and perhaps starts with scratch elements), the “Same layer” setting gives the desired results.
- **D Code Size Rounding**

In “Setup/D Codes” (F5), the “Operations” menu has a new rounding item. In addition to the existing “Round to closest mil” (or “Round to closest 0.01 mm” in metric mode) there is now “Round to closest 0.1 mil” (or “Round to closest 0.1 mm” in metric).
- **“Select/Large Pads”**

Will select pads (in visible layers) which have at least one dimension larger than a user given value. This is useful for selecting pads that are really large copper areas (not component landings). These pads can then be replaced with polygons so that they do not interfere with netlist export and other commands that work specifically on pads (like delete unused pads).
- **“Edit/Edit Selection/Paint And Scratch/Raise To Top”**

A new command that deals with the paint/scratch order of the selected elements. This command will change the polarity order so that the selected elements will be above (on top of) all the unselected elements (layer by layer). All paint elements in the selection remain paint and all scratch elements remain scratch, just that they will come earlier in the paint/scratch order when all elements in a layer are considered. They will be “promoted” on top of everything else and hence guaranteed to be visible.
- **“Edit/Edit Selection/Polygons/F To Polygons”**

A new option has been added to facilitate processing of complicated drawings. It is “Allow Intersecting

Holes". This option is only relevant when "Nest Polygons" is checked. It allows for intersecting holes and will merge them to make combined holes.

- **"Tools/Netlist/Import Copper Polygons"**

This command is available only after a contour netlist extraction has completed. It will import (into empty layers) polygons for the relevant copper layers used in the netlist extract. The polygons will have the correct net numbers. It is often more convenient to examine connectivity using these polygons (especially for negative layers and layer with filled areas) because there are fewer elements to consider and (in the case of negative layers) elements do not represent copper and so will not get net numbers.

- **"Setup/Frame/Surround All Data" (Ctrl+W)**

A new option has been added "Consider scratch elements also". When unchecked, scratch elements will not be considered in the computation. When checked, scratch elements are considered the same as if they were paint (this was the previous behavior of this command).

- **Mapping D Codes to N-type shapes**

The user interface for re-mapping non-N shapes to N-shapes in "Setup/D Codes" (F5) has been improved.

- **"File/ Import/ ODB++"**

Importing component and Netlist data has been re-enabled.

- **"Tools/Netlist/Extract Contour"**

The adjacency computation is now significantly faster.

9.8.50:

- **New Layer Types**

The following new layer types have been added:

PTH	Drill – Plated Through Hole
NTH	Drill – Non plated Through Hole
BLI	Drill – Blind vias
BUR	Drill – Buried vias
ROU	Routing – generic
RPE	Routing – external plated
RNE	Routing – external non plated
RPI	Routing – internal plated
RNI	Routing – internal non plated
PLA	Plating
SCG	Scoring

- **"File/Import/Gerber" - "Options" – "File Interpretation"**

⇒ **"Zero-Length Draws"**: an additional option has been added: "Ask". Previously there were only two possibilities: "Make pads" and "Ignore". Now there are three: "Make pads", "Ask" and "Ignore".

"Make pads"	A pad will be generated to replace the zero-length trace
"Ignore"	Zero-length draws will be ignored
"Ask"	The user will be prompted and given one of the above choices

⇒ **"End of Block Characters"**: an additional option has been added: "Auto". When "Auto" is selected, the software will try to determine which the proper setting is for the file being

imported.

- **“View/Grid/Display Grid” (G)**
The grid points are now rendered with 5 pixels. The previous 1-pixel grid was difficult to see on high resolution monitors.
- **“File/Import/Guess- Directory- Zip”**
 - ⇒ HPGL- recognition has been improved.
 - ⇒ IPC-D350/ IPC-D356 – now searches in the comment areas to determine type.
 - ⇒ **“Tool diameters use opposite units”**- check-box has been added to the results dialog for drill files. When checked, the tool codes will use the opposite UOM than the data (inch – mm, mm – inch). To use this feature in import directory and Zip, highlight the file in the scan results then press “Edit Format”.
- **“Tools/Convert Drawn Pads/Single Shape” (Ctrl+Shift+P)**
Polygons are now supported. They will be interpreted as zero-width contours for the draw to flash operation.
- **“Tools/Netlist/Extract Contour”**
The two separate dialogs (extract and export) have been combined into one tabbed dialog.

9.8.20 / 25

- **New Selection Criterion**
A new entry has been added to the selection criteria in the Selection Toolbar. It is called “Scratch Elements” and, when active, will only select/pick elements that have scratch polarity inside a layer. Note that if the whole layer is set to scratch, that will not cause elements in it to be picked.
- **“Setup/Preferences/Splash Screen”**
Allows enabling or disabling the display of the splash screen at program startup.
- **“Tools/Compare Layers/Save Differences To File”**
The coordinates where differences were detected are saved to a text file.
- **Character Coding for Import and Export**
The character coding (ASCII, EBCDIC, EIA) is now separate for import and export. Previously changing one would also change the other.
- **“File/Import/Guess Directory Zip”**
 - ⇒ TAP format drill files with T-Code definitions in the header section will now use the T-Code value rather than the hole size number. The ‘n’ value still defaults to three. The ‘m’ value is now calculated by subtracting the ‘n’ value from the total number of digits in the coordinate data.
 - ⇒ The results dialog now resizes better.
 - ⇒ Gerber files that use CR/LF instead of ‘*’ are now recognized.
 - ⇒ Recognizing HPGL and HPGL/2 files has been improved.

9.8.5:

- **“Select/Covered Active Elements”**
Now handles zero-width elements also. This includes traces mapped to F-type D Codes.

- **“Edit/Edit Selection/Polygons/F To Polygons”**

Algorithm has been improved.

9.8.0:

- **Inserting Scratch Elements Directly**

One can now insert scratch elements into any layer.

Previously, if one wanted to add new scratch elements, one needed to add them to a scratch layer (the whole layer was marked as scratch). While that is still possible, there is a new way to insert scratch elements: the commands “Paint” and “Scratch” in the “Insert” menu. There is also a “scratch” button on the Insert toolbar, which will cause scratch elements to be inserted when the button is depressed (active). While scratch mode is active using any of the above commands, any of the elements inserted with the normal insert commands (“Insert/Pad”, “Insert/Trace”, etc.) will be scratch elements. More precisely, scratch elements will be inserted into positive layers and paint elements into negative layers, the end result being that they will scratch away from copper.

Note that the “Only” selector next to the layer box in the “Current Element” toolbar will be taken into account (as always the case with insert operations). So, if you want to insert into one layer only, make sure that the “Only” selector is checked.

- **Selecting A Particular Element When There Are Multiple Stacked Elements**

Sometimes there are multiple stacked elements in the same layer. Often this is the case with stacked pads (perhaps a scratch pad to create a clearance and then a paint pad). Since they have the same center, it was previously difficult to select a desired element (just the scratch or just the paint in the example above).

There is now a new way to do this. By executing the select command repeatedly, the selected element will cycle through all candidates in order and one can stop when the desired element is selected. This means double-clicking repeatedly (or single-clicking repeatedly when the selection tool is active).

- **Routing Paths Compensation**

Compensation is now supported for **open** paths also. Previously only closed paths could be compensated.

- **Layer Colors**

There are new commands for saving and restoring the layer color setup. The color setup consists of the color index for each layer (a number between 1 and 15, which maps to one of the 15 colors that can be set at any time) and the color composition (RGB) of each of these 15 colors.

These commands are **“Setup/Layer Colors/Save To File”** and **“Setup/Layer Colors/Restore From File”** and they allow saving and restoring the configuration.

To make such a setup the default setup (loaded when the program starts) the command **“Setup/Layer Colors/Save As Default”** should be used.

Also, the complete color setup is now saved in .bin files. Previously only the layer indices were saved.

- **Distance Measurement**

There are several changes and new features in the distance measurement commands (**“View/Distance”**).

- ⇒ The distance is now displayed only in the view (next to the cursor). The Screen Toolbar will display relative X and Y displacement from the start of the distance measurement. The screen display also includes the angle from the start of measurement.
- ⇒ The “distance from edge” measurement has been merged into the distance tool. Is it now an option that can be turned on and off (**“View/Distance/From Edge”**), so there is only one distance tool with two modes.
- ⇒ The command to measure the edge-to-edge distance between two selected elements has been replaced with a tool (**“View/Distance/Edge To Edge Measurement Tool”**). While this tool is

active one can click on two elements (click on first, then click on second) and the edge to edge distance between these two elements is displayed in the status line.

- **Transcoding NC Tools By Descending Size**
The tools setup dialog ("Setup/Tools Codes" or "Shift+F5") had two buttons for transcoding tools by ascending size. These have been combined into one button "Transcode By Size" with two more methods added to allow transcoding by descending tool sizes also (previously only ascending order was implemented).
The four methods are: "Ascending", "Ascending By Type", "Descending" and "Descending By Type".
- **"Setup/Preferences/Center On Wheel Zoom"**
Controls how the zoom happens when zooming with the mouse wheel. When this option is active (checked) the zoom is combined with a pan so that the position where the zoom started becomes the new center of the view.
- **"View/Coordinates Origin/Set Relative Origin"**
This is a newly added command that sets the relative origin at the current cursor position. It takes the name of the previous "set relative origin tool" (and also the shortcut Shift+R) which is now renamed "Relative Origin Tool" and has no shortcut key. However, it keeps its button on the Screen Toolbar.
- **"File/Import/File Path Options/Current Directory Is Set On Import"**
This is an options that can be turned on or off. When on, the current directory is set to the path of the file-name imported. This can be desirable if one imports files by dragging and dropping (rather than using the File/Import dialog).
- **"File/Import/Drill"**
A new option (in the "Options" dialog "Data Format" tab) called "Tool diameters in opposite units". When checked, the tool diameter values read in will be interpreted in the opposite units from the ones specified in "Units". The value in "Units" will be used to interpret coordinates. If the file contains unit directives, such as M71, M72, INCH or METRIC, they will of course override any settings from this dialog.
This is useful for importing files which have the header and the rest of the data in different units, such as diameters in metric and coordinates in English or vice versa.
- **"File/Export/Gerber Extended"**
Two new options have been added in the "Options/RS274-X Options" tab. These will warn the user when the file to be output will contain apertures that are either zero-width or wider than a user specified value. This is useful when outputting to photoplotters (such as the Gerber Crescent) which have aperture size limitations. These options are: "Warn for zero-width D Codes" and "Warn for D Codes of diameter larger than".
- **"File/Export/Drill"**
A new option has been added for exporting Sieb & Meyer files. It is in "Options/Output Options" and is a pair of M commands that can be output at the top and bottom of the file. These commands are for machine specific M commands, such as M38 (jam) and M39 (loosen), but any numbers can be entered. Sieb & Meyer describes the "Machine Function" commands as being M22 through M27 and M34 through M41. If a box is left empty, then no extra command will be output at that level (top or bottom of the file).
- **"Tools/Design Rules Check/Add Errors From File"**
New command similar to "Load Errors From File" but does not clear the existing errors first. This way one can merge the results saved in several error files into one combined set of errors.

- **D Code rotation**

The operation to rotate a group of D Codes, available in the “Setup/D Codes” (F5) table, has been changed to allow for a user specified angle of rotation. Previously only a rotation of 90 degrees was available. The command is now “Operations/Rotate” (was “Operations/Rotate 90 Degrees”).

- **“View/Distance/Edge To Edge”**

Has been changed from a command to a tool. See description under “Distance Measurement” in the “New” section below.

- **“View/Coordinates Origin/Set Relative Origin”**

This command has been renamed “Relative Origin Tool” and has lost its shortcut (Shift+R) now linked to a new command that sets the relative origin but is not a tool (immediate action). See below under “New” commands.

- **Improvements In Contouring Algorithm (Polygon Merging)**

This algorithm is used in many places, but most importantly in Contour DRC and Contour Netlist extraction. The speed of the algorithm has been dramatically improved for cases where the data to be merged results in a crosshatch. This can often happen in plane layers. Previously such layers were best handled with Raster methods, but now Contour methods can be used throughout.

- **“File/Save Options/Preferences”**

The “Include Net Info” option has been removed from the dialog. Now net info is always saved in a job-file (.bin file), if it is present (if there are elements in the job which have a net info value different from “?”).

- **“File/Import/Gerber/Options”**

The “Data Format” tab items “Left of decimal” and “Right of decimal” are selectively grayed out to convey to the user when they are irrelevant. “Left” is irrelevant in “Leading” omit zeros mode and “Right” is irrelevant in “Trailing”.

- **“File/Import/Guess”**

The Import Guess results dialog has changed for setting m and n. It now uses two buttons to shift the decimal point instead of dropdown boxes. Also the “Apply” button has been removed as the board size is automatically recalculated whenever a parameter is changed.

- **“File/Import/Apertures”**

When importing Pads aperture files, German headers will be recognized. Also the units will default to mm.

- **“File/Import/ODB++”**

- ⇒ The ODB++ layer name value will be used during import to set the layer name in the F10 menu.
- ⇒ ODB++ will now tag the first copper layer as CPU, last copper layer as CPL. Any layers before the CPU that are silk, soldermask, or solderpaste will be marked as ‘U’pper. If these files are listed the CPU layer, they will be marked as ‘L’ower. Drill layers are now DRI instead of PAD.

9.6.91:

- **“View/Visibility/Layer/Board #0 Always Visible”**

A new option for controlling the behavior of the commands that step the visibility of layers. These are in “View/Visibility/Layer”: “Next Layer”, “Previous Layer”, “Next Board Layer” and “Previous Board Layer”.

When “Board #0 Always Visible” is checked, then these commands will always cause board layer #0 to stay on (visible) together with the next board layer in the sequence (next or previous).

- **“File/Import/Net”**
Better handling of IPC-356 “307” records (blind and buried via).
- **“Edit/Edit Selection/Polygons/Make Well Formed”**
The algorithm has been improved to handle a larger variety of malformed polygons.
- **“Select/Duplicate Active Elements”**
Elements that have no width will now be processed by this command also. Previously they were ignored. This includes pads and traces mapped to D Codes that have zero width or of shape type “F”.
- **“Tools/Netlist/Netcompare”**
The comparison algorithm has been improved.
- **“File/Import/DPF”**
Now uses the Windows temp directory for temporary files during import.

9.6.81:

- **“File/Import/Apertures”**
A new aperture type: Proteus.inf is now supported.
- **“File -> Import -> Directory/ Zip”**
A new button has been added to the scan results dialog, “ReScan”. When pressed, the user can change the import option settings and rescan the files. In the past the user had to start the import over from the beginning.

9.6.78:

- **“Edit/Edit Selection/Special/Convert Routing Paths To Traces”**
This command replaces the selected routing paths with traces, one trace per edge. Previously this command was available only in CAMMaster (and higher) as “Tools -> Routing -> Ungroup To Traces”.
- **“Edit/Edit Selection/Special/Convert M-type Pads To C-type”**
This command acts on the selection and replaces each M-type pad in the selection with a pair consisting of a polygon and a C-type pad. The polygon has the same shape and position as the original M-type pad so that the copper is preserved. A C-type pad is also placed on top of it, within the bounds of the polygon.
The main use for this is for preparing data for netlist generation and for software that cannot deal with custom (M-type) pads.
- **“File -> Import -> Apertures”**
ApSource7 now imports diamonds.
- **“File -> Import -> ODB++”**
Import time has been greatly improved.

9.6.69:

- Drag The View Using Scrollbars
When “View->Preferences->Drag View In Real Time” is checked, dragging the scrollbar boxes will also drag the view in real time. Note that the “Drag View In Real Time” also controls the behavior of dragging the view with Ctrl+Alt+Left Mouse button (this was present in previous versions also).
- “Setup->Tool Codes” (**Shift+F5**)
A new command has been added for transcoding tool numbers. It is called “Transcode Selected” and will transcode all the selected tools (which can be contiguous or not) to a contiguous range of tool numbers specified by the user. Of course one can transcode just one tool number, for instance T4 to T9, but also sets like T1, T4, T7 to start at T10, which will cause the transcoding: T1 to T10, T4 to T11 and T7 to T12.
- “File->Import->”From Layers” Uses Cur Dir”
A new setting which applies only to the command “File->Import->From Layers”. This command imports several files using the layer names as file names. For names that are not full paths, it is not obvious how to complete the path (which directory they are in). Previously the directory was assumed to be the directory of the jobfile. When this new option (“From Layers” Uses Cur Dir) is checked, the “current directory” is used instead. The current directory is an operating system dependent concept and it is changed when file browsing dialogs are used. It is usually the last place where a file was saved/exported or opened/imported.
- “Paint And Scratch->Expose Scratch”
Command available in the context menus in the layer toolbar and layer setup dialog (F10). Will correct layers that start with scratch elements (which are invisible). It will add a covering paint rectangle in front of them so as to expose the scratch elements.

9.6.53:

- Layers Toolbar
A new way to quickly turn on or off a range of layers has been added.
This is done by clicking the right mouse button and pressing either the Shift or Control keys at the same time. If neither of these keys are pressed, then the normal context menu is displayed. However, if any of these keys is pressed, this action will cause instead to change the visibility of the layers ranging from the current layer to the layer on which the right button is clicked. Shift will cause the range to be made visible, Control will cause the range to be made invisible.
- “Help/User ID”
Shows the current User ID for this product and allows the user to enter a new one (in case the existing one is incorrect). Previously the User ID could be only set during installation.

9.6.43:

- “File/Import/Drill & Rout”
A new option has been added for importing Excellon files. It is in the “Options” tab of the “Options” dialog. It is called “M00” and controls how M00 statements are interpreted.
There are three possibilities:
 - ⇒ “Treat as end of program”. This will cause the current program to be terminated and any following data will appear on a new layer.
 - ⇒ “Ignore”. The statement will be completely ignored.
 - ⇒ “Increment tool”. The statement is interpreted as a tool change to the next sequential tool number.

- **Crash Report Generation**
If the software crashes, a dialog will come up asking whether to email a crash report to PentaLogix technical support. These reports will be investigated by technical support and they help to find and fix the source of the problem.
- **"File/Import/Aperture"**
A new aperture type "Metric Source" has been added. This aperture type recognizes files that use commas instead of decimal points and has special conversions for an aperture named "SYMB" to targets.
- **"File/Import/Guess" (also, Directory and Zip)**
Gerber extended (RS-274X) files with incorrect format statements (FS) are now easier to import. Hard to believe, but some Gerber files have format statements which are incorrect. Previously the only way to correct this was to edit the file and change or remove the statement. Now one can use "Import/Guess" (or "Zip" or "Directory") which will guess the format and will allow the user to override it in the dialog that appears (check-box item "Override parameters").
- **Pads / Traces Contrast**
Contrasting pads and traces with different colors in the same layer is not guaranteed to display correctly when the board layer contains paint and scratch. However, certain users need this feature so much that they would rather use it on such layers also, even if the display is not guaranteed to be correct. This version enables them to do so, by checking the option "Allow contrast on paint/scratch also" in "View/Visibility/Element/Options". This is a "use at your own risk" kind of option. The data will always be correct, but the paint/scratch rendering may be incorrect when pads contrast is on.
- **"Edit/Paint And Scratch/Optimize Selection"**
The optimization operation has been improved to pair scratch and paint polygons. Polygons with holes are often represented via paint and scratch because some of the import languages do not have ways of representing polygons with holes directly. Gerber is such an example where holes are represented as scratch polygons. The optimization operation will now detect such situations and add the scratch polygons as holes of the corresponding paint polygons. This results in fewer scratch elements and sometimes in the complete elimination of scratch elements.
- **"File/Import/Apertures"**
If the aperture import encounters problems, an error file is generated. When the dialog box appears to report that errors have been encountered, a button has been added to view the error file.
- **NC Tool Sizes**
Tool sizes are now saved in jobfiles (binfiles) even if they are not mapped to any D Codes. This facilitates using common setups of NC tools. One can set up the tools, then save an empty (no circuit data) binfile, which can later be opened or imported to restore the settings.

9.6.17:

- **"File/Import/Gerber/Options/RS274-X Options"**
⇒ The handling of "IPNEG" commands has been improved. These are used to define a layer as a negative layer. "IP" stands for "Image Polarity". The issue is how to treat subsequent "LP" commands ("Layer Polarity"). These can be "LPD" or "LPC". Some files assume "LPD" to be the paint polarity, some "LPC" ("LPC" seems to be more prevalent). Previously there were only two modes for this option. Now a third one, called "Automatic" has

been added that will try to guess the correct setting.

9.6:

- “Edit/Move Layers/Align Visible Layers” (Ctrl+Q)

A new command to facilitate aligning layers has been added. It incorporates the methods that were available in the old “Edit/Move Layers/Align” (which it replaces) and it adds more methods.

This command works on all visible layers and will try to move some of them so they align with the others. The following methods are available:

Pad or trace in a corner of a layer. The corner to be considered can be specified by the user (upper-right, lower-right, upper-left or lower-left). The software will pick (in each visible layer) the element (pad or trace) closest to the corner in that layer and will use them as reference elements. The layers will be moved so that the reference elements end up on top of each other.

One selected element in each layer. The selected elements will be used as references in each layers. The layers will be moved so that the selected elements end up on top of each other. If more than one selected element exists in a layer, an error will be reported and the operation aborts. Layers which have no selected elements will be ignored. To select elements, one can use the selection tool (S) or simply double click on an element. Remember that to add to an existing selection one needs to keep the Shift key pressed while selecting.

One selected pad whose type is used as a guide. This is the old “Edit/Move Layers/Align” that has been around for quite a while. It requires the selection to consist of just one pad. If pads similar to it (using the additional rules of “D Code Number Only”, “Shape Type Only” or “Shape Type And Size”) are found in the other layers, the closest to the selected one is picked in each layer and used as the reference for that layer. The layers are then moved so that all these pads end up on top of each other.

This works well only for cases where the layers are not offset by much and there are distinguished pads (such as targets). If all layers have targets, aligning targets is an easy way to do the job.

In each of these cases some layers will be moved and others not. One layer (at least) is always left where it is. For method (3), that layer is the layer that contains the selected element. For methods (1) and (2), the largest number of layers that already match are left standing. If no layers match already or the operation is done on only two layers, then the “current layer” is left standing.

For many cases more that one method needs to be used. Pads can very often be used for alignment, but some layers have no pads (such as silkscreen layers) and so a second step to align them may be necessary (using traces, or selected elements).

- “Select/Covered Active Elements”

A new command that will first clear the selection and then select only the active elements that are completely covered by other elements. These are usually redundant, garbage elements and can be discarded. Note that if two (or more) identical elements are superimposed in the same layer, all will be selected, except one, so that deleting the others does not modify the copper of the layer. Candidates for selection are only active elements, so they must be visible and also all the “Only” buttons and controls are taken into account. One could, for instance, apply it just to pads.

- “Select/Duplicate Active Elements”

A new command that will first clear the selection and then select only the active elements which are identical (geometrically) to others. These are usually redundant, garbage elements and can be discarded. Note that of two (or more) identical elements all will be selected, except one, so that deleting the others does not modify the copper of the layer. Candidates for selection are only active elements, so they must be visible and also all the “Only” buttons and controls are taken into account. One could, for instance, apply it just to pads.

This replaces the older command “Select/Duplicates” which acted a bit differently, working only on the

elements already selected.

- **“File/Import/Drill & Rout”**
Added support for importing Excellon commands G32 and G33 (“Routing Circle Canned Cycle”) and Sieb & Meyer commands G45, G46 (“Rout A Hole”), G47, G48 (“Rout A Round Card”) and G49, G50 (“Rout a Hole (Rectangle)”).
- **“Setup/D Codes” (F5)**
A new command for selecting elements mapped to a set of D Codes. It is in the menu of the “D Code Table” dialog under “Operations/Select Pads And Traces”. For all the picked D Codes in the table (which are highlighted in dark blue) the elements mapped to those D Codes, and in visible layers, will be selected. This allows selecting elements mapped to more than one D Code.
- **“Tools/Netlist/Select Net”**
A new selection tool that facilitates selecting all elements having the same net name (or number). This was possible previously also, but required a sequence of keystrokes to do, as the “Only” button next to the Netlist box in the Current Element toolbar needed to be turned off then on repeatedly.
This is no longer needed with this new tool. Once activated, one can click (or hit Enter) to select the active element closest to the cursor and, together with it, all the other active elements which have the same net number. This tool acts like the selection tool in the sense that the Shift and Ctrl keyboard keys change the meaning of the selection: if Shift is pressed while clicking, then the operation will add to the selection, if Ctrl is pressed, then it will remove from the present selection and if neither is pressed, then it will make a new selection.
- **“File/Export/Rout/Options”**
A new page has been added to the options dialog which allows arcs to be converted to chords for export purposes. This is for routing machines which cannot handle arcs. The routing paths in the database are not modified.
- **“Tools/Routing/Display X’s At Plunge Points”**
When turned on, will display an X over the plunge points of currently visible routing paths. These are the points where the routing paths start.
- **“Setup/Preferences/Only One Tool Can Be Active”**
This is a new preference, but it modifies a behavior that existed previously. Certain user tools: zoom (“View/Zoom/Tool” (Z)), select (“Select/Selection Tool” (S)), distance (“View/Distance” (D)) and others are considered “helper” tools in the sense that it is sometimes useful to activate them while another (perhaps multi-step) tool is active (in dormant state). When the helper tool is canceled (by Escape or otherwise) the dormant main tool is reactivated.
Some users do not want this behavior, because they would rather have just one tool active. For those user, this new option has been introduced. When checked and a helper operation is activated, any previous operation is canceled.
- **“File/Import/ODB++”**
 - ⇒ Improved import algorithm is much faster.
 - ⇒ The import process has been streamlined by importing according to the ODB matrix rather than requiring the user to select import order.
 - ⇒ Step and repeat, rotations, and offsets are now imported correctly.
 - ⇒ The profile step is now imported as a BOL type.
 - ⇒ When running in Batch mode, all menus are suppressed.

- **“File/Import/DPF”**

When importing files in metric, the conversion is now much more accurate by using a conversion factor to eight decimal places.

- **“File/Import/Gerber”**

- ⇒ The settings in a Gerber extended “FS” statement (Format Statement) will not change the user settings in File/Import/Gerber/Options. They will of course apply to the current plot, but will not affect later files. This allows the values set by the user to persist.
- ⇒ Improved handling of approximate arcs (when the radius at one endpoint is different from the radius at the other endpoint).
- ⇒ The “AM” Rotation modes for “Rotate around” have been expanded to three. Previously there was “Rotate around center of each primitive” and “Rotate around aperture center”. Now a new mode has been added, called “Automatic”. This will pick one of the two previous modes, depending on the type of primitive.
This has been determined after examining a number of sample file, some of which needed both types of rotation in the same file.
The auto mode seems to be working well with all the sample files examined, but the other two modes have been left in the user interface as a fallback position, to account for different types of Gerber files that the user may encounter.

- **Selection Toolbar**

There are several new options to specify elements to select or pick (in the dropdown list):

- ⇒ “Open Paths”: picks open routing paths only
- ⇒ “Closed Paths”: picks closed routing paths only
- ⇒ “No Lead-In”: picks closed routing paths which have no lead-in
- ⇒ “Net Midpoints”: picks elements which have a net number (or name) and have been marked as net midpoints by a previous net import or net extract operation.
- ⇒ “Net Endpoints”: picks elements which have a net number (or name) and have not been marked as net midpoints by a previous net import or net extract operation.

- **“Edit/Break Traces And Clip Polygons” (Ctrl+B)**

This command was called “Break Traces” before. Now polygons are included in the operation also (when a frame is used for clipping). Any polygons that touch the frame will be clipped by it and subdivided into several polygons, similar to the way a trace is clipped. This provides a powerful new way to manipulate polygons.

Note that this operation is affected by the state of the “Inside” button on the “Selection” toolbar.

- **“File/Import/Guess-Directory-Zip”**

- ⇒ The **import** options now has check-boxes to include/ exclude certain file types. The file types are:
 - ⇒ Scan for Data Files
 - ⇒ Scan for Aperture files.
 - ⇒ Scan for Netlist files
- ⇒ The **default** import order has been changed. Drill files are placed near the bottom of the list; and drill files with tool definitions but no data, immediately precede the drill files. Netlist files are placed last. This is to accommodate net lists with ‘T’ codes.
- ⇒ **Certain** file types are filtered from scanning to prevent crashes. Files ending in “PDF”, “DWG”, and DXF” are ignored. Other files are scanned to detect binary data and will be ignored if appropriate.
- ⇒ Zip **now** has the same options button found in import directory.
- ⇒ Tap files without format data will default to the most common parameters. This file format will now be identified as three possible types:

Tap definitions without tool changes will be treated as an aperture file, no drill data imported.

Tap files with Excellon tool definitions will be treated as Excellon drill files.

TAP files with TAP tool definitions and with tool changes will be identified as a new type, NC_TAP. Tool definitions and data imported.

- ⇒ Imports both the tool and drill data if present. The file type is called “NC-TAP/ Excellon”. When the format isn’t provided, default settings will be used that seem to work in most cases.
- ⇒ Recognition of Excellon format improved handling of ‘T’, ‘F’, ‘C’, ‘S’ commands. Metric 1, 2, formats recognized.
- ⇒ Gerber files with large AM (Aperture Macros) import quicker.

- **“File/Import/Apertures”**

- ⇒ Valor aperture files now import diamonds, thermals, obrounds, and octagons.
- ⇒ LAV501 apertures now supports octagons and tool definitions.

- **“View/Distance”**

The distance measurement tools now display the current distance next to the cursor in the main view, in addition to the X box in the “Screen” toolbar.

- **“Help/Check For Updates”**

The user can now specify an interval of time for automatically checking for software updates from the PentaLogix website. This check will be done in the background and the user will only be notified if there is a new version available.

- **“Tools/Snap To Padmaster”**

The “Snap Settings” tab has a new option “Select result”. This determines whether the elements that were snapped are selected or not at the end of the operation. The previous behavior was that they were always selected.

- **“Tools/Compare Layers”**

The speed of the algorithm has been improved.

- **“Tools/Netlist/NetCompare”**

The “Select” button has been removed from the errors display floating dialog.

Selection is now done automatically. When a new error is examined all the nets that appear in that error will be selected. Further selection refinement can then be done by the user, by clicking on the net names in the NUT and REF lists. The Ctrl and Shift keys can be used to change the selection in the normal Windows convention, to modify an existing selection or to select ranges.

9.4.73:

- **Unmapped D Codes**

Unmapped D Codes (of shape type “?”) and N type D Codes for which the corresponding tools are unmapped (of type “?”) will be displayed as a question mark inside a doughnut. This is to make them easy to recognize when looking at the display. Previously they were displayed as rounds.

- **“File/Import/Zip”**

Now has the options button available when selecting a file to import. This is the same options button found in Import Directory.

- **“File/Import/Directory” and “File/Import/Zip”**

The options dialog now has three new check-boxes When selected, File import will scan files for these

data types:

- ⇒ Scan for Data Files.
- ⇒ Scan for Aperture files
- ⇒ Scan for Netlist files

- “File/Export/Drill”:

A warning is issued when trying to export data that contains unmapped tools (of type “?”).

9.4.66:

- “Tools/Compare Layers”

The “Comparison Method” options have been implemented. There are two methods for comparing two layers:

- ⇒ “Element by element”

Each element in the first layers is compared with a corresponding element in the second layer. Where differences are found, elements are selected.

- ⇒ “Copper area”

The copper are of the first layer is compared with the copper are of the second layer and a marker will be displayed at spots where there are differences (copper in one layer but not in the other). These differences can be navigated via a dialog. If one closes the dialog the markers will be removed, but they can be brought back again with (“Tools/Compare Layers/Show Differences”).

This method is also different from the “Element by element” method in that it handles composites (step and repeat blocks and text blocks).

9.4.6 - 65:

- “File/Import/Discard Path In Layer Names”

Controls how layers are named from file names during import operations.

When checked, only the file name part will be stored as the layer name and the path part of the file name will be discarded.

When unchecked, the hole path name will be used, unless the file is in the same directory as the currently loaded jobfile.

- “Setup/Visibility and Display”

“Step & Repeat Blocks”. New options have been added for the display of step and repeat blocks. The two options are “Display all details” and “Display outlines only”. The new one is the second option. If selected, it will cause all step and repeated data to be displayed on the screen as a boxes, without the detail of the circuit elements inside. This saves time when examining large panels with a lot of panelized data. Note that this applies only to “blocked” step and repeat data. If the data was step and repeated using copying, this option does not apply, as the step and repeat information is not recorded in the database.

- “Setup/Layers” (F10)

New commands to save/restore the default color setup for layers display. The commands are available from the context menu (right mouse click) under **“Colors Setup”**. There are three commands:

- ⇒ **“Save Current As Default”**: the current setup of layer colors and color definitions (RGB content of colors) becomes the default. The next time the software is started, it will display these settings as the default.
- ⇒ **“Restore From Default”**: layers colors will be set to the default settings.
- ⇒ **“Restore From Factory Settings”**: layers colors will be set to the PentaLogix default values.

- “Edit/Edit Selection/Move By Displacement”
A new command to move the selection by user specified X and Y displacement values.
- “Insert/Pad”
During the insert operation an outline of the shape to be inserted is dragged along with the cursor, giving better feedback as to what the result will be. This includes outlines of complex shapes also.
- “View/Distance”
The distance measurement tools (“From Point (D)” and “From Edge”) now stay active until explicitly canceled. So they will stay active when starting other tools, such as the insert trace and pad tools. This is useful for accurate positioning during such insert operations.
- Pad outline mode (O)
Custom apertures (shapes of type M) will be showed outlined with primitives displayed individually (both paint and scratch primitives are shown). This allows for a better understanding of the components of a custom shape.
- “File/Import”
 - ⇒ “Gerber”
“AM” rotation. It has been determined that both interpretation for the rotation of AM apertures exist in Gerber files. The two possibilities are:
 - rotate each primitive individually around its center or
 - rotate the whole aperture (around the aperture center)
 In versions previous to 9.4, the second method was used, then in the early 9.4 releases it was change to the first method. We have now seen files that assume one or the other, so the decision was made to add a user setting, so the user can pick which one to use. It is in “Options/RS274-X Options” and it is called “Rotate around”.
 - ⇒ “Bitmap”
Better handling of color bitmaps based with a color palette.
 - ⇒ “Directory/ Zip/ Guess”
Recognition of Excellon format has been improved.
The Excellon format statements Metric 1-3 is now recognized.
Adobe Acrobat .PDF, Illustrator .AI, AutoCAD .DXF and certain .DSN files will be ignored for import.
The import order has been changed. Netlist files will be listed last. Drill files will be listed before the Netlist files. If a drill file does not contain coordinate data, it will be the first drill file listed.
These changes will help drill sizes to be properly imported.
 - ⇒ “ODB”
ODB import is now much faster. Step and Repeat as well as rotate is now handled properly.
Profile layer is now imported as BOL. The menu to arraign layer order has been eliminated. The layers will now import in the order defined by the ODB file.
- “Setup/D Codes/Operations/Unique Shapes”
A new option has been added called “Shape types to consolidate”. This allows handling N-type shapes differently. One can apply the operation only to N-type shapes, only to non-N type shapes or to all shapes.
- “Tools/Compare Layers”
Two new options have been added:
 - ⇒ “Stop after ... errors”. Allows the user to specify a number of errors after which checking should stop.

⇒ “Tolerance”. Differences smaller than this will be ignored.

9.4.1:

- **“File/Import”**

- ⇒ **“Gerber”**

- The extended Gerber command “IR’ (Image Rotation) is now supported.

- ⇒ **“Drill & Rout”**

- Sieb & Meyer rout data is now supported. Previously only drill data was supported for Sieb & Meyer.

- **“File/Export/Rout”**

- Sieb & Meyer rout data can now be exported via the “Sieb & Meyer” command.

- **“Insert/Arrow”**

- A tool to insert arrows made out of traces, similar to the arrows generated by “Insert/Dimension”, except that there is no dimensioning. These arrows can be used to mark locations of interest. The height of the arrow is derived from the “Text Height” parameter in the “Inset/Loc and Dim Setup” dialog, similar to the “Insert/Dimension” tool. Once the tool is active, you need to click first at the position of the head of the arrow and then at the position of the tail of the arrow.

- **“Select/Layers”**

- A new command to select all elements in a group of layers, specified as a comma-separated list of layer ranges (for instance “3-5, 12, 14-22”). This will make a new selection, any previously selected elements will be deselected first.

- **“Tools”**

- ⇒ **“Stencil Apertures”**

- A new set of tools for creating stencil apertures. The commands and features are described in a separate document: “Stencil Apertures”.

- ⇒ **“Routing/Arc Chamfers”**

- Acts on selected routing paths. Adds arc chamfers of the specified radius to vertices where two straight edges meet.

- **“Netlist/Extract Contour”**

- ⇒ **Polygon Output**

- Polygon data is now output in F04 netlist files. This allows polygons to be displayed in the test programs to improve viewing/ inspection.

- ⇒ **Adjacency and Shielded Adjacency**

- Option to find shielded adjacencies for faster testing using flying probe testers as defined in the IPC-356B standard. You can choose to determine shielded adjacency for all nets or to ignore blobs (large drawn areas). If “ignore blobs” is selected, the blobs will be treated as normal adjacencies. The shielded adjacency is only exported in F04 format.

- ⇒ **New midpoint algorithm**

- Option to identify midpoints. You can choose to identify midpoints on all netlists or to ignore blobs (large drawn areas). The latter option is recommended for faster completion and if it is alright to treat all the pads on the blobs as endpoints.

- ⇒ **F04 C\$J record**

- A new record C\$J for the adjacency threshold is output for the F04 netlist output format. The format is: C\$JXXX where X is the distance in 10th mils.

- **“AOI/CAMTEK”**
The output of data to CAMTEK AOI inspection machines has been finalized and tested.
Please see the document “CAMTEK Interface” for a description of features and procedures.
- **Component Information**
Component information is now stored in the database when available from the input files. Currently component information is imported from netlist files (columns 21-31 of IPC-356) and ODB++ files. It is stored as system custom properties called “_Component_Name”, “_Component_Type” and “_Component_Pin”. These can be used via the custom properties toolbar and the new “Select/Components” command and can be qualifiers in selection operations. One can, for instance, select all elements that have a given component name. More information on this can be found in the “Stencil Apertures” document, although component information can be used independent of stencil apertures.
- **“View/Distance”**
The distance measurement tools (“From Point (D)” and “From Edge”) now stay active until explicitly canceled. So they will stay active when starting other tools, such as the insert trace and pad tools. This is useful for accurate positioning during such insert operations.
- **Pad outline mode (O)**
Custom apertures (shapes of type M) will be showed outlined with primitives displayed individually (both paint and scratch primitives are shown). This allows for a better understanding of the components of a custom shape.
- **“Setup/D Codes ” (F5 dialog)**
 - ⇒ The dialog has been redesigned to allow for column resizing. One can do this in the usual way by dragging the gridline that separates two headings in the headings row, for instance between “X” and “Y”.
 - ⇒ One can sort the table by values in a given column. This can be done by clicking on the heading of the column. For instance, clicking on “X” will sort the table by the X dimension of the D Codes (ascending or descending).
 - ⇒ The dimensions of the D Code (in the X and Y columns) are displayed at their highest accuracy. Previously, only 6 digits were displayed and the value was rounded if it needed more than 6 digits (including the decimal point).
 - ⇒ The shape of the current D Code is displayed in the dialog.
 - ⇒ The whole table can now be printed: “Operations/Print”.
 - ⇒ The “similar shapes consolidation” command “Operations/Unique Shapes” will now consolidate two D Codes of N-type that are mapped to different tool codes but the same diameter.
- **“Remember Directory for Import”**
Has been replaced with “Remember Directories”. When checked, separate directory names are remembered for import, export and macros.
- **“View”**
 - ⇒ **“Visibility/Layer/Toggle Layers”:**
A comma separated list of layers or layer ranges can now be entered. Previously only a single layer could be toggled.
 - ⇒ **“Toolbars/Macro/Custom”**
The maximum number of buttons has been increased from 20 to 30.
- **“Insert”**
 - ⇒ **“Loc and Dim Setup”**

A new option has been added for the “Insert/Location” and “Insert/Dimension” tools: “Append Units”. When checked the text for the units (“inch”, “mm”, etc.) will appear after the dimension number, otherwise only the number will appear by itself. Previously the text for the units was always present.

⇒ **“Pad”**

During the insert operation an outline of the shape to be inserted is dragged along with the cursor, giving better feedback as to what the result will be. This includes outlines of complex shapes also.

- **“File/Import”**

⇒ **“Zip”**

Now reports CRC errors when attempting to unzip files. Also applies to TGZ files for ODB++.

⇒ **“ODB++”**

The net information will now also be imported, if present in the ODB file. If multiple netlists are available, the user will be prompted to choose the appropriate netlist. PentaLogix layer type recognition has been improved. ODB can now be imported with messages suppressed for batch mode operations

⇒ **“DPF”**

TEXT commands are now handled. Nested blocks now supported. Rotations, scaling, etc. are supported.

⇒ **“Netlist”**

Board outline records (389) are now recognized and are imported as “F” shape traces.

⇒ **“Apertures”**

Added a new GAPFile type. Supports the following string types:

!GAPFile Version 3.0

11 Round d:8.0

309 Oblong w:13.0 h:48.0 a:0.00

310 Rectangle Sq w:84.0 h:76.0 a:0.00

- **“File/Export/Rout/Excellon”**

A “Data Format” tab has been added to be “Options” dialog. This allows setting the format of the output file. Previously the data was always output in “Absolute” with “Explicit decimal point”.

- **“AOI/Lloyd Doyle/Contour”**

Lloyd Doyle now outputs the max X and Y scan coordinates instead of the can area. This will correctly place the test data on the tester. The Lloyd Doyle dialog now allows the user to enter values in Inch, Mil, CM, or MM.

- **“Tools”**

⇒ **“Generate Polygon Outlines/Vector”**

A new selection has been added to the “Outline Settings” tab. It specifies what the resulting outlines will be: polygons or contours made of “F” shape traces. In older versions of the software “F” shape contours were generated, then this was changed to polygons in more recent versions and finally now both methods are available, as many users seem to prefer the “F” shape contours.

⇒ **“Netlist/Extract Contour”**

Vias are marked with a ‘P’ plated in the Hole Definition Field (column 38) for both F04 and IPC netlist file formats.

⇒ **“Routing” - Display of NC routing paths**

The display of the plunge point in a routing path has been modified. Previously a marker was displayed when a path was selected, but that led to confusion as the marker could be left behind without any direct connection to newly displayed data. Now the plunge point is showed as a line

perpendicular to the path and surrounded by a circle of diameter equal to the tool size for any displayed path. This display exists only in outline (zero-width) mode (which can be toggled with "O").

9.2.41:

- **"Tools/Routing/Set Precedence"**

A new tool to allow quick setting of routing path precedence for the selected routing paths. While this tool is active a blue box will be displayed at the plunge point of all paths currently in the selection, with the precedence numbers displayed inside the boxes. The precedence numbers of this group of paths can be modified by clicking over the boxes in the desired order. The starting precedence number (the one assigned on the first click) is the smallest number existing in the group when the tool is activated. If Ctrl is pressed while clicking, the numbers are decremented rather than incremented. This allows for quick correction of inadvertent clicks.

- **"File/Export/Rout/Excellon"**

An Options dialog has been added. There is currently one option which controls how compensated paths are exported. The user can choose whether the export is done using Excellon G commands (G41 and G42) or whether the path is displaced by the compensation amount and G commands are not used.

- **"File/Print"**

The print dialog has two new items: a "Font" button and a "Raster" check-box

"Font" allows selection of the font for the header. Previously a default font was used and will still be used if no new font is picked.

"Raster" picks which printing method is used: raster bitmap based or Windows GDI based. Note that for Windows ME and lower only the raster method is available. For the other Windows versions the user can pick the preferred method. The Windows GDI based method is faster and works with all printers, whereas the raster method (which sends a potentially large raster bitmap to the printer) is more similar to the screen display (in color mode) as translucency of colors is supported (although the colors may be different on screen and on the printer).

- **"Tools/Routing"**

⇒ **"Properties Tool"**: If **Shift** is pressed when the tool is activated (via mouse click or pressing Enter), instead of having the properties dialog pop up, only a message describing the path is displayed on the status line. This is intended to allow quick inspection of a number of paths without having to dismiss the dialogs that pop up.

⇒ **"Properties Tool"** and **"Selection Properties"**: a diameter box has been added to the dialogs so that the diameter of the tool can be quickly viewed and changed.

- More robust handling of network (floating) licenses.

- **Import Apertures / Netlists**

In any dialog where the user has the option to view a file, the users' defined viewer will be used. If the file type doesn't have a viewer assigned, wordpad.exe will be used.

9.2.27:

- **"File/Import/ODB++"**

By selecting "From Dir", the user can now import ODB jobs directly from the directory without having to create a TGZ or TAR file.

- **“Edit/Edit Selection/Polygons/Separate Holes”**

The holes that result from the separation will be selected, whereas the containers will be unselected. This makes it easy to further process holes if desired. Previously both the holes and the containers were selected.

9.2.21:

- **“Tools/ Routing”**

The properties for routing paths have an added element: compensation. It can have three states: None, Left and Right. With no compensation (None), the path is viewed and exported in its present form. When compensation is on (Left or Right), then the path will be offset by the radius of the tool so that the original path represents what is left after cutting. Currently only closed paths can be compensated.

- **“Layers Toolbar”**

When a layer contained only scratch elements inside (although the layer as a whole could be marked as paint) it was difficult to figure out what was going on, as the layer would be invisible when viewed by itself. Now, layers of this kind are marked with a “+” (instead of the “*”) to draw attention that something unusual is going on. Then the right click context menu command “Paint And Scratch/Force Inner Polarity To Paint” (which replaces “Separate Layer...” in this case) will make all the inside elements paint and also flip the polarity of the whole layer (if it was paint it becomes scratch and vice versa). At this stage, the data is equivalent to what was there before, but now the polarity can be easily manipulated (via the “Scratch” check-mark).

- **Mouse Wheel Behavior**

Moving the mouse wheel scrolled up and down when the scrollbars were visible and zoomed in and out otherwise. The behavior when the scrollbars are off is unchanged. However when the scrollbars are on, moving the wheel will scroll up and down when no other key is pressed, scroll left and right when the Shift key is pressed and zoom in and out when the Ctrl key is pressed.

9.2.2:

- **“Select/Duplicates”**

Looks for duplicates among the currently selected elements. Only the duplicates will remain selected while all the other elements are deselected. Duplicates are elements that are geometrically identical and in the same spot (on top of each other) and of the same shape (although not necessarily the same D Code). Currently only pads and traces are checked for duplicates (polygons and routing paths are not).

- **“Edit/Edit Selection/Special/Merge Colinear Traces”**

Looks for linear traces in selection that continue each other and are colinear (on the same line) and replaces them with one trace which spans the whole extension.

- **Layers toolbar context menu (right-click)**

Five more commands have been added under “Select”: “All”, “Traces”, “Pads”, “Polygons” and “Routing Paths”. These commands will select only elements from the layer that was picked and the selection is made independent of the other criteria (the “Only” buttons etc.) as well as independent of element visibility. It is always a “new” selection, meaning that all other previously selected elements will be deselected. The previously existing commands “New”, “Add” and “Remove” are preserved and work the same way as before (they are affected by the normal selection criteria).

The new commands give a quick way of selecting a whole layer or just some particular element type within a layer without worrying about the other settings in the software.

9.2:

- Paint elements and scratch elements **can now coexist in the** same layer.

Previously a layer was paint or scratch and all elements inside were of the same polarity (paint or scratch). However, because some import files contained many polarity switches (from paint to scratch and from scratch to paint) each requiring a new layer, the result was a large number of layers. It could even happen that the maximum number of 255 layers was exceeded and so the file could not be imported. Starting with this version, a polarity change in the input will not require a layer change as paint and scratch elements can now coexist in the same layer.

To see how many polarity changes there are in a layer, make that layer the current layer and then do “File/Properties” or Alt+Enter. The number of polarity changes is one of the information lines in the displayed dialog. Multi-polarity layers are also marked with a ‘+’ in the layers toolbar (instead of the ‘*’ for a single polarity layer).

One can still set the polarity of a whole layer as paint or scratch (same as before). If the whole layer polarity is scratch then the polarity of elements inside will be reversed: paint elements will act like scratch and scratch elements will act like paint.

A layer which contains both paint and scratch elements can be separated into an equivalent sequence of layers which contain only one polarity. This makes it easier to edit the data (if necessary) and also conforms to how the software acted in previous releases (for users who prefer it that way). Note that for each polarity switch an extra layer is required. This can be done using commands in the “Paint And Scratch” submenu which appears in several places:

- ⇒ in the main **menu** under “Edit”
- ⇒ in the layers **toolbar** context menu (right-click of mouse)
- ⇒ in the Layers **Table** dialog (F10) context menu.

There are two commands for separation:

- ⇒ “Separate Current Layer”
- ⇒ “Separate All”. This separates all layers that have mixed polarity. Note that for users who prefer the old way of storing paint and scratch, the operation to do is “Separate All” after each import.

The reverse operation also exists, to combine a sequence of layers which belong to the same board layer (and perhaps have different polarities) into one layer. Commands for this are in the same “Paint And Scratch” submenus:

- ⇒ “Combine Layers”
- ⇒ “Combine All”. This looks for adjacent layers that belong to the same board layer (and are **visible**) and combines them **into** one layer. The operation is then repeated for all the layers in the job.

The only way to change the polarity of elements is by setting the layer polarity. So if one needs to change the polarity of only a group of elements inside a layer, one needs to transfer them to an empty layer immediately following the layer in question, set the board layer number to be the same, then change the layer polarity to the desired value and finally combine the two layers into one layer again.

- **A new version of binfiles** is also introduced with this version (V10 of binfiles)

This is only needed when saving jobs that have mixed polarity elements in the same layer. Note that older versions of the software will not be able to read/open these binfiles.

- **“Paint And Scratch”**

The menu under “Edit” has one more command that acts on the selected elements only: “Optimize Selection”. This command will examine the selection, layer by layer, and within each layer will attempt to reduce the number of polarity switches. Sometimes input files contain a lot of polarity switches that are unnecessary because later paint and scratch elements do not interact (overlap) with previous ones, so it is possible to merge them into fewer polarity groupings. This can be useful when one needs to edit the data and also makes the export/output more efficient.

- **“Insert/Chamfer”**

Activates a tool to insert, move (slide) or delete a chamfer trace. This tool is a combination insert and edit tool. While the tool is active, pressing “Shift” puts it in the insert mode, while pressing “Ctrl” puts it in the remove mode. If none of the keys are pressed, it is in edit (move) mode.

A chamfer trace is a trace connected at both endpoints to other linear traces which act as “rails”. The chamfer can be moved parallel to itself while being guided by the rails. It can be a linear trace or an arc trace. In the edit/remove mode this is determined by what trace is picked for editing, in the insert mode this is determined by “Insert/Arc Chamfer Mode” (once set, it will be remembered for future operations). Note that when picking (either the chamfer to edit, or the junction of the rails where to insert) only active elements will be considered (this means all the “Only” buttons are active).

The insert operation will create a new chamfer starting at the joint of the two rails, while the remove operation will delete an existing chamfer and extend the rails so that they meet.

These operations can be used to rearrange circuits so that they provide more clearance spaces where needed.

- **“Edit/Edit Traces”**

A new group of operations to edit traces. They affect up to three traces at a time. The traces can be straight (linear) or arc traces. The following table provides a summary of the operations.

<i>Name Of Operation</i>	<i>Action</i>	<i>Traces affected</i>
Move Trace	Picks closest trace. The trace then follows the cursor by moving parallel to itself.	1
Move Endpoint	Same as “Move Trace”, but operation starts by picking trace at closest endpoint only.	1
Move Vertex	Picks closest junction (vertex) of two traces. Both traces then follow the mouse by moving parallel to themselves and staying connected.	2
Move Chamfer	Picks closest trace. Each endpoint of the trace needs to be connected to another trace. All three traces then follow the cursor by moving parallel to themselves (as a group).	3
Drag Trace	Picks closest trace and traces connected to each of its endpoints. The picked trace follows the cursor by moving parallel to itself while the connected traces will be stretched (the other endpoints of the connected traces will stay unchanged).	3
Drag Endpoint	Picks the closest trace endpoint. The endpoint then follows the cursor while the trace is stretched (the other endpoint stays unchanged). NOTE: this is the same as the older “Edit/Stretch Traces”.	1
Drag Vertex	Picks closest junction (vertex) of two traces. This vertex then follows the cursor while the traces are stretched (the other endpoints stay unchanged).	2
Drag Chamfer	Picks the closest trace and the two guiding rails attached to each endpoint. The trace then follow the cursor while guided by the rails. NOTE: this is similar to the “Insert/Chamfer” tool in edit mode.	3

The shortcut key for the whole group of commands is F8. Pressing F8 will activate (or cancel) the last used “Edit Traces” command. Pressing Shift+F8 and Ctrl+F8 will step to the next or previous tool (up or down through the menu listing).

- **Viewing of selected elements:**

Two new ways of visualizing selected elements have been added to the two that existed. They can be picked from “View/Visibility/Selected”, “Setup/Visibility” or the Visibility Toolbar. The four modes are:

<i>Mode Name</i>	<i>Color For Unselected</i>	<i>Color For Selected</i>
Display In White	Layer color	Bright white
Display In Contrasting Color	Layer color	Color that contrasts with layer color
Display By Dimming Unselected	Dimmed layer color	Layer color
Display Unselected In Gray	Dark gray	Layer color

The last two modes are the new ones. In both of these modes the selected elements are displayed in the layer color (both traces and pads, even if contrast is turned on). This is meant to help in cases where the continuity of the selected elements across layers is important to visualize, such as when examining netlists. In the last mode the unselected are displayed in one color independent of what layer they belong to. This color defaults to dark gray but can be set by the user to any other color using the “Set Layer Color” dialog that can be reached from several places (“Setup/Colors”, right-click on the Layers Toolbar or from the “Setup/Layers” dialog).

- “Edit/Edit Selection”:
 - ⇒ **“Replace With Polygons”**
Replaces all selected elements with equivalent polygons. This is similar to the existing “Tools/Generate Polygon Outlines” (“/Vector” for CAMMaster) except that it acts on the selection only and does not swell the resulting polygons during the operation.
 - ⇒ **“Polygons/Replace With Overlaps”**
A new command that acts on the selected polygons (other elements in the selection are ignored). This operation takes every pair of polygons in the selection and replaces them with their overlap (or intersection), which is the copper area common to both polygons.
Note that if two polygons do not overlap, there will be nothing to replace them with, so polygons in the selection which do not overlap any polygons will be deleted. Please see the document “Polygon_Intersection.doc” for more information.
 - ⇒ **“Special/Replace M-Type Pads With Primitives”**
This is a new command which works on the selection. It will replace all pads in the selection that are of shape M (Custom) with the primitives that make up the shape. The result will look exactly the same, but will contain no custom shapes. This can be useful when data needs to be exported to devices (or software) that have trouble understanding custom shapes (photoplotters etc.). A similar command already existed in the software, but was only accessible from the D Code Table (F5) and did not act on the selection, but rather on the D Codes that were picked from the table. That command was “Custom Apertures/Replace With Primitives” and is still there in this version as well. It has been improved to handle custom shapes that contain scratch primitives. Previously such shapes were not processed. Now, these shapes are converted to polygons and then the pads are replaced with these polygons.
“Replace M-Type Pads With Primitives” works the same way: shapes with scratch primitives are replaced with polygons and shapes without scratch primitives are replaced with the constituent primitives (pads, traces, polygons).
- **“File/Import/Drill & Rout”**
Support for FMAT command in Excellon program header.
- **Blankouts**
Blankouts (also known as Exclusion Zones) are a set of rectangles that can be specified by the user. They are not part of the database and will not be exported or imported and in that way are similar to the frame. The main purpose of blankouts is to mark areas that will be excluded from certain operations. Currently they only apply to netlist extraction, Lloyd-Doyle AOI output and CAMTEK AOI output, but may in the future be used for other operations as well.
The “Blankouts” menu appears in several places: in “Setup”, in “Tools/Netlist” and in “AOI”.

When “Blankouts/Insert” is selected, the user can add new blankout rectangles by clicking and dragging with the mouse or using the “Tab” key from the keyboard (same as one would operate for selecting). Blankouts can be saved and read from files. Two formats are available: the PentaLogix format and the CAMTEK format. Blankouts are also saved in jobfiles (.bin files).

They can be turned on and off with the “Blankouts/Show” command. Blankouts affect the netlist and AOI operations in the following way:

- ⇒ Contour netlist (“Tools/Netlist/Extract Contour”): blankouts act as scratch elements during netlist extraction. They will erase anything that they cover. This can be useful when extracting a netlist after certain fabrication artifacts have been added that would alter the connectivity information in undesired ways. Plating bars across fingers is an example. One can tell the netlist extract to ignore them by placing blankouts over them.
- ⇒ Contour Lloyd-Doyle (“AOI/Lloyd Doyle/Contour”): blankouts will act as described above, but only when “Blankouts/Treat As Scratch” is checked. Otherwise (if unchecked) the blankouts will have no effect during netlist extraction.

In both cases, whether checked or unchecked, if blankouts exist and are visible (“Blankouts/Show” is checked) then the blankout info will be output to the Lloyd-Doyle file and will be available at the Lloyd-Doyle workstation.

- **“Setup”:**

- ⇒ **“Layers”** (F10) Context Menu: a new command has been added – “Compact”. The purpose of this command is to eliminate any empty layers from within the sequence of layers in a job. Layers will be transferred so that the relative layer order is the same, but there are no intervening empty layers.
- ⇒ **“Visibility”:**
All the global visibility settings for elements have been assembled into this dialog. All these items exist as other menu entries or dialogs, under “View/Visibility/...” but have been assembled here for quick access.
- ⇒ **“Colors”:**
To set the color of the current layer.

- **Context Menu (mouse right click):**

Added a new command: “Draw To Flash” which will perform the same operation as “Tools/Convert Drawn Pads/Global” with the currently set parameters (it will not display the dialog for the parameters). The method used is “standalone”.

- **“Tools/Delete Pads”:**

The “delete unused pads” part of the “Delete Pads” module has been significantly improved. The new code uses contouring methods to find the unused pads, whereas the old method used vector methods. In addition to being more accurate, the new method handles: arcs, polygons, custom shapes (M-type D Codes), paint-and-scratch and negative layers. None of these were handled by the old method. The old method is still available under “Tools/Delete Pads/Vector”. The new method is “Tools/Delete Pads/Select Unused Pads”.

It works on the selection only, so you need to select all the elements (both pads and non-pads) that you want considered. In addition, as for all other operations, only visible parts of the selection are processed. Within the selection, the processing is done one board layer at a time, with proper consideration for paint-and-scratch as well as negative layers.

Unused pads are those pads that are electrically connected only to other pads (or are completely isolated). Used pads are those that are electrically connected to traces, polygons or to negative copper areas.

At the end of the operation the selection will consist of the unused pads ONLY. If there are no such pads then the selection will be empty. At this point these pads can be easily deleted (by pressing the Delete key) or they can be further processed as desired.

- **“Tools/Routing”:**
Two new commands have been added which operate on selected routing paths: “Arcs To Chords” and “Chords To Arcs”. They are similar to the commands which operate on polygons, but work with both open and closed routing paths. The “Chords To Arcs” command which will try to approximate sequences of edges with arcs, can be used indirectly to convert to arcs chains of traces also. Several operations are needed in this case: select the traces in the chain, do “Tools/Routing/Group To Paths” to convert to routing paths, then “Tools/Routing/Chords To Arcs” and finally “Tools/Routing/Ungroup To Traces” to get back the traces.
- **“Layers Toolbar”**
Two buttons have been added to facilitate access to two frequently used commands from “View/Visibility/Layer”: “Next Layer” (Shift+Equals) and “Previous Layer” (Shift+Backspace).
- **“File/Import”:**
 - ⇒ **“Excellon”:** Files with ‘T’ codes that also have ‘Feed or ‘Speed or ‘Diameter commands on the same line will be recognized as Excellon.
 - ⇒ **“Apertures”:** TraxMaker apertures are now supported using the same algorithms as Protel.
 - ⇒ **“Directory- Zip- Guess”:**
Gerber files that use CR/LF as a block delimiter. Dialog boxes now have radio buttons for Asterisk or CR/LF block delimiters.
G70/71 (inch/mm) will be recognized in extended Gerber files.
Will now gray out parameters defined in files so that the user cannot change things like the Inch/MM.
Will gray out Quadrant and 360 arc buttons only if G74/ G75 have been found, and G74/ G75 appear in the file before the arc commands (G02/ G03).
Excellon files in inch will be set to 2,4 but won’t be grayed out in the view format dialogs.
When “Format unknown files to known files” is checked in the options box; if the unknown file had m,n or in/metric defined, the algorithm will not change those values.
- **“File/Print”**
A new item has been added for the “Content” box: “Visible layers (combined)”. This will print on one page all visible layers.
- **“Swell”:** speed has been improved, especially for swelling large polygons.
- **“Polygons”:** “Make Well Formed” and “Chords To Arcs”: More accurate results are computed.
- **“Setup/Layers”**
The height of the rows in the layers dialogs is now remembered and restored. The height can be adjusted by moving the cursor to the border of two rows, waiting for the cursor shape to change, then dragging the border to the desired position.
- **Visibility Toolbar**
A new button has been added for the visibility of polygons. A group of four buttons have been added to control how the selected elements are displayed (two of these modes are new in this version, see below).
- **Insert Toolbar**
A new button has been added for the new “Insert Chamfer” tool.
- **“AOI/Lloyd Doyle”**
Blankouts are supported. See the description of blankouts under “Added features”.

In addition the frame interacts with the export of Lloyd-Doyle data in two ways:

- ⇒ The lower-left corner of the frame will be the origin (0, 0) of the exported data.
- ⇒ Only elements inside the frame will be exported. Elements that are completely outside will be ignored and elements crossing the frame will be clipped at the frame border. Note that connectivity will be computed considering all elements (even the ones outside the frame). It is only at the export stage that the frame interacts with the data.

- **“Tools/Netlist/Convert Netlist File/To IPC”**

PADS Netlist Import now recognizes drawn pad shapes such as circle, square, and rectangle. The output will now generate all square and rectangle shapes rather than outputting those shapes as circle. Inner layers are now included in the output.

9.0.50:

- **Context menu** (the right-click menu over the view)

The most recently used command will appear as the first entry of the context menu, to facilitate repeating the same command.

- **“Edit/Edit Selection/Arcs/Break Into Quadrants”**

Same as “Tools/Convert Arcs/Break Into Quadrants”, but works on selected arcs.

- **“File/Import/Drill & Rout”**

“Options/Options”: new options added for importing Excellon files.

The purpose of this new option is to specify the user desired handling for the “Block Delete” command (the slash (“/”)) at the start of a block. There are now three possibilities in the new “Block Delete” box:

(1) Follow BLKD command: the BLKD command in the header of the file will specify whether blocks marked with slash will be read in or ignored. The two forms of the BLKD command are “BLKD,ON” and “BLKD,OFF”.

(2) Read in always: BLKD commands in the header are ignored and blocks marked with slash are always imported.

(3) Delete always: BLKD commands in the header are ignored and blocks marked with slash are always deleted (ignored).

Note that the last two options will override any BLKD setting in the header of the Excellon file.

- **“File/Save Options”**

The two options: “Include Net Info” and “Step And Repeat Blocks” have been moved to a new dialog: “Preferences”. This dialog also has a new item, the desired version number of the jobfile (.bin file).

With newer versions of the CAM software, as more features were added, the jobfile format needed to be updated. The current version of the jobfile format is 9, but it is in general not directly related to the version number of the software. Older versions of the software can of course not read these new jobfiles, so the option has now been added to save older version jobfiles when needed.

When a user chooses for output an older version jobfiles and the job contains elements and features that cannot be represented in that version, those elements will be dropped or converted (if possible). The user will be prompted before the output.

For a short history of jobfile version numbers press the “Info” button in the “Preferences” dialog. This feature is especially useful when saving files for the Pulsar photoplotters. The Pulsars cannot accept files newer than version 6.

- **“File/Export/Cutter Code (HPGL)”**

There is now an “Options” dialog that allows specification of whether to output pads and the manner in which they are output. There are three choices: pads are not output, they are output as a “Pen Down” or

they are output as a small circle. Previously pads were always ignored.

- “File/Import/Guess or Zip or Directory”
Filenames with the extension of “.err” will be ignored. Files that have import parameters defined in the file, will be displayed with those parameters grayed out (cannot be changed).

9.0:

- Basic support for routing
New data structures have been added to represent NC routing paths. Simple rout files can be exported and then processed further in RoutMaster. The new commands are found under “Tools/Routing”. Excellon rout files can be imported. For more information on this topic please see the document “CAM Support for Routing”.
- Custom Properties
These are properties that can be attached to the database elements: pads, traces etc. They have the form “Name=Value”, where Name and Value are arbitrary. Their greatest use is as a selection criterion. That way one can first select (and then operate on) elements that have certain custom properties. There is a new toolbar “View/Toolbars/Custom Properties” and a new command which appears in two places: “View/Selection/Custom Properties” and “Select/Custom Properties”. For a more detailed description please see the document “CAM Custom Properties”.
- **A new layer type has been added**
The new type is a generic plane layer type. Previously there were five power layer types (PW1-PW5) and five ground layer types (GD1-GD5). In addition to these, there are now 99 generic plane layer types (P01-P99). They can be used for netlist extraction for either ground or power plane layers.
- **A new navigation aid has been added**
One can now drag the view by dragging the mouse (with the left button pressed) while both the Ctrl and Alt keys are pressed. Since redraw times can be long when a lot of data is loaded, the drag can be done in real time or not. This real time feature can be enabled or disabled via “Setup/Preferences/Drag View In Real Time” or “View/Preferences/Drag View In Real Time”.
- Area Computation.
One can now compute the copper area of the selection with a command that appears in two places: “View/Selection/Area” and “Select/Area”. It is similar to the “Tools/Calculate Copper Area” in that overlapping elements will be counted as combined and that scratch elements will subtract from the area computation. It acts only on the currently selected elements and will display the result of the computation in a dialog so that the resulting number can be pasted into other documents. Similar to other tools and commands it currently operates only on primitive elements. This means that composites: step-and-repeat blocks and text blocks are not counted.
- **Commas as decimal separators**
In the case of numbers typed into dialogs: commas can now be used as decimal separators (as well as points) to facilitate data entry in countries where the point is not the decimal separator. So the numbers “1.2345” and “1,2345” will be considered the same.
- “File/Import”:
⇒ **“Guess”**: a new command for importing files of unknown type or format. The software will attempt to guess the type (Gerber, DPF, Excellon, netlist, etc.) and (where applicable) the format (Leading/Trailing etc.). The file is then imported into the current layer.

- ⇒ **“Aperture”**: the wizard now has an additional radio button: **“TCodes”**. When selected, only a tool code and a drill size need be identified. The wizard will translate the codes into Tool codes rather than D codes.
- ⇒ **“Directory”** and **“Zip”**
Now recognizes bitmap files (.BMP), and netlist files.
- **“Insert/Text”**
 - ⇒ A new check-box has been added to the **“Add Text”** dialog called **“Grouped”**. This will determine whether the newly inserted text will be grouped or ungrouped. It was done so that users who prefer the old (ungrouped) way of storing text do not need to do an **“ungroup”** after the text is inserted (one can always do an **“ungroup”** after the fact using **“Edit/Edit Selection/Text/Ungroup”**).
 - ⇒ A new font has been added: **ODB Standard**.
- **“Edit”**:
 - ⇒ **“Edit Selection”**:
“Replace Arcs With Chords”. Same as **“Tools/Convert Arcs/Approximate By Chords”** but works on the selected elements only. Note that the **“Tools...”** function works on all active elements that are inside the frame.
 - ⇒ **“Polygons/Replace With Skeleton”**:
Each selected polygon will be replaced by a set of F-Type traces and pads which constitute the **“Medial Axis”** or **“Skeleton”** of the polygon. The skeleton is the set of all points inside a polygon such that a **“clearance disk”** centered at such a point touches the polygon at two (or more) distinct places. Note that pads can also be part of the skeleton: the skeleton of a polygon that is a circle will be just one point and so a trace cannot be used to represent it (it would be zero-length). There are three methods for generating the skeleton:
Complete: the whole skeleton is generated
Backbone only: small details around junctions points are eliminated.
Endpoints only: this is not really a skeleton, but pads are generated at the **“endpoints”** of the selected polygons.
 The last two methods use a parameter called **“Brush Width”** which specifies the width of the details that can be ignored. A brush of the given width, painting along the backbone, should never paint outside the boundaries of the polygon.
 - ⇒ **“Polygons/Replace With Inscribed Pad”**:
Each selected polygon will be replaced by a circular pad, such that the circle that the outline of the pad represents is the largest possible inscribed circle in that polygon. A larger circle cannot be fitted inside the polygon without crossing the edges.
 - ⇒ **“Step And Repeat”**: A new way to load the step distance. There is now a button to the right of the Step X and Y fields (labeled **“<”**). When pressed it will load the step fields from the current values of the X and Y boxes in the **“Screen”** toolbar. Most useful when the display is in relative mode or distance measurement mode.
- **“Setup/Preferences”**:
 - ⇒ **View In Real Time**: see the **“drag the view”** item above.
 - ⇒ **“Ignore Preferences From Jobfiles”**: affects the processing of user preferences and parameters when job files (binfiles) are opened. When checked, these preferences will be ignored (the data in the job files is of course read in). Some users prefer to have the preferences set a certain way and do not want them changed by opening a job file. One example of a set of such preferences are the Gerber import options (from **“File/Import/Gerber/Options”**).

- **“Setup/D Codes”**

A new command to quickly go to the first unused D Code following the current D Code (“Goto/Next Unused”).

- **“Setup/Frame/Visible”**

Command to turn frame visibility on or off.

- **“Setup/Advanced/Intensity Contrast By RGB”**

This defines how the color intensity contrast is done for the following commands: “View/Visibility/Element/Options/Show Pad-to-Trace Contrast As Contrasting Intensity” and “View/Visibility/Selected/Display In Contrasting Intensity”:

When unchecked, the contrasting color is taken from the table of fifteen defined colors. The table is conceptually divided in two halves, where the lower half (colors 1-7) is usually the “low intensity” half and the upper half (colors 8-15) is usually the “high intensity” half. The contrasting color will be the corresponding spot in the other half (so 1 and 9 are contrasting, 2 and 10 etc).

Note that the actual colors in the table of 15 can be redefined by the user in an arbitrary fashion, so the two halves may not really reflect the “low intensity”/“high intensity” division, but the contrasting mechanism will stay the same.

When checked, the contrasting color will be derived by flipping the high order bit of each R,G,B component of the layer color. For instance, the color that is normally color “1” in the fifteen color table (Dark Blue) has R=0, G=0, B=127 (7F hex), so the contrasting color will be R=128, G=128, B=255 (a color that is not in the default fifteen color table).

Some users prefer this mode, although it can be problematic. For instance: a color very close to the Dark Blue from above, for instance R=0, G=0, B=128 (80 hex) will contrast to R=128, G=128, B=0, which is quite different to the previous contrasting color.

- **“Location and Dimensioning Tool Setup” dialog**

A new mode for the number of digits after the decimal point has been added: “Exact”. When this option is chosen, that exact number of digits will appear after the decimal point including any trailing zeros.

This is useful when the number of digits after the decimal point imply a tolerance for the drawing. The old mode of “Maximum” still exists and works the same way – at most that many digits will be generated, but trailing zeros are dropped.

- **“Go/Favorite Positions”**

A new command has been added, “Remove All”, to clear the current list of favorite position (leaves the list empty).

- **“Help”:**

⇒ **“Check For Updates”**: Checks the PentaLogix website and reports if there are newer versions of the software available for downloading.

⇒ **“Release Notes”**: To read the release notes for the current version.

- **New accelerator keys (shortcut keys):**

File/Import/Drill	Shift+F2
File/Import/Guess	F4
File/Import/Directory	Ctrl+F4
View/Toolbars/Navigation	Shift+F12

- **“View”:**

⇒ **“Toolbars/Editing”**: the **“Delete”** button has been moved to the end of the toolbar. Some users

have complained that it was too easy to press it by mistake.

- **“Edit/Edit Selection/Swell”**

Previously, if N-type D Codes were in the selection, an error message was displayed and the swell was not done. Now, N-type D Codes in the selection can be swelled, but only if all of the D Codes of that tool number are in the selection (otherwise a transcoding of the tool number would be necessary and this is currently not supported).

- **“Insert”:**

⇒ **“Text”**: the font of the dialog has been changed to Courier, which is a fixed width font. This facilitates entering tabular data into the “Text” box (columnar text where the columns need to line up). Note that for this to work the chosen CAM software font should also be of fixed width (one of: DMPL, Uppercase, ODB, 4X7 or 5X7).

⇒ **“Location”**: the coordinates that are inserted will be relative to the “Relative Origin” when in “Relative” mode.

- **“File/Import”: “Directory” and “Zip”**

the transcoding report (if there is one) is written to the directory of the imported file(s), with name “Transcode Report.txt”. Previously the user settable report file path was used (from “File/Report/Set Name”).

- **“Tools”:**

⇒ **“Generate Outlines/Vector”:**

All D Code shapes are now processed correctly. In the past, more complex shapes were approximated by rectangles.

The result of the outlining operation is polygons, instead of F-type traces. If the F-type traces are the desired result, the polygons can be converted to that by using the “Edit/Edit Selection/Polygons/Polygons To F” command.

Important note for those who use the Macro command “.GenerateOutlines”: you will need to change your code to convert the polygons to traces (as mentioned above) after calling “.GenerateOutlines”.

⇒ **“Convert Drawn Pads/Global”:**

speed has been improved and results are more accurate.

8.4.50

- **Layer Visibility:**

“View/Visibility/Layer” Two new commands have been added to facilitate turning ON and OFF groups of layers:

⇒ **“Make Visible”** - Shift+F10

⇒ **“Make Invisible”** - Ctrl+F10

One can type a list of layer range, such as “2-4, 14, 20-23” and this list can be interpreted as either layer numbers or board layer numbers (determined by the “Board Layer” item in the dialog).

- **High Resolution Only**

“File/Save Options”: The “High Resolution” option has been removed. Binfiles are now always saved in high resolution. Low resolution is only needed for versions 6.0 or earlier of the CAM software.

- **Scroll Bars Changed**

“View/Scroll Bars”: the scroll bars now scroll over the loaded data rather than over a user defined work-

space ("Setup/Preferences/Workspace"). For instance: moving the horizontal scroll bar to the leftmost position will place the view over the left margin of the loaded data.

- Contour Netlist Enhancements

"Netlist Extraction/Contour" IPC and F04 output files have been enhanced:

⇒ Net name (fields 4-17) are now left justified

⇒ Board Outline is now included:

as 389Board_EDGE (board outline record). Any trace data in CAMMasters' BOL layer will be transferred.

8.4.10

- Custom Toolbar for Scripts:

For user scripts (Sax Basic macro files) "View/Toolbars/Custom"

Each button is mapped to a file, which must be a Sax Basic compatible Basic file (".bas" extension). The buttons will display a bitmap or a button number. A bitmap will be displayed if a ".bmp" file exists in the same directory as the ".bas" and has the same name as the ".bas" file. These files can be made by the user using Microsoft paint or some other application that handles Windows bitmap files. The dimensions of the bitmap should be 16 by 16 pixels. One can add buttons to the toolbar by right-clicking on the toolbar and selecting "Add A New Button" or by selecting one or more ".bas" files in the Windows Explorer and dragging and dropping them on the toolbar. With the second method several buttons can be added at once. One can delete buttons unless the toolbar has only one button (there is always at least one button on the toolbar). After adding buttons to the toolbar, it may need to be resized so all the buttons are showing.

- Elements Overlapping Cursor :

"View/Info/Elements Overlapping Cursor": A new command to provide information about elements covering the cursor. A list of all the elements (pads, traces etc.) which cover the cursor will be displayed.

This is useful when investigating pad stacks, as it is difficult to know how many pads there are on top of one another. It can also be used in troubleshooting connectivity as one can see the net names of all elements that connect at a certain point.

- Exporting Polygon Data:

"File/Export" Gerber varieties: Improved rounding of polygonal data when exporting to lower resolutions (lower than .6 in English or .5 in metric).

8.4

- Navigation View:

A navigation view has been added to show an overview of the relative positions of the current view and the visible data and to allow quick modifications of the view ("View/Toolbar/Navigation"). The current view is represented as a black rectangle and the extents of the visible data are shown as a filled green rectangle.

As opposed to the other toolbars, the Navigation View is not dockable, as its square nature would cause it to take up too large an area of the screen. However its size and position can be modified by the user and it is remembered and restored at the next activation of the software. The input focus can be shifted to the Navigation View by clicking on it or by pressing Ctrl+PgUp. While in the Navigation View Ctrl+PgUp will transfer control back to the main view.

The Navigation View window responds to the following user interaction:

⇒ **Left mouse button drag-and-draw:**

the view will match the user drawn rectangle.

⇒ **Left mouse button click:**

will center the view at the spot of the click (at the current zoom level).

- ⇒ **PgUp:**
zoom up (same as pressing PgUp in the main view).
- ⇒ **PgDn:**
zoom down (same as pressing PgDn in the main view).
- ⇒ **Arrow keys:**
pan the view in the direction of the arrow.
- ⇒ **Home:**
Same as pressing Home in the main view.

- **Wheel Mouse Support**

Added support for navigation with a wheel mouse and support for the middle button of three-button mice:

- ⇒ **Rolling the wheel**
zooms in and out. Rolling forward zooms in and backwards zooms out. Each click zooms the same amount as a PgUp or PgDn would do. This mode is only active if the scrollbars are off. When the scrollbars are on, rolling the wheel will normally scroll up and down.
- ⇒ **Clicking the wheel**
(or middle-button of a three-button mouse) will pan the image so that the spot where the click happened will end up in the center of the screen. This is similar (though not identical) to using the End key. One can pan *repeatedly* in one direction by making sure that the arrow and crosshair cursors are not joined, then moving the arrow cursor close to one of the window edges and clicking the wheel repeatedly.

Notes about setting up the mouse: Mouse button actions and properties can be set up in the Windows Control Panel. The wheel is often set up to cause a scrolling action and pressing the wheel (or middle button) is sometimes set up to simulate a left button double-click. These are user preference that can be changed if desired. This is how these settings interfere with the new features introduced above:

- ⇒ **Scroll:**
If the wheel is set to scroll (in the Control Panel) and the CAM scrollbars are visible, then moving the wheel will scroll vertically rather than zoom. This action was present in older versions of the software as well. Note that if the scrollbars are not visible (“View/Scroll Bars”) then the setting in the Control Panel does not matter.
- ⇒ **Middle Button:**
If the wheel (or middle button) click is set to some other action (other than “Middle Button” or default), the panning described above will not work. Instead, pressing the wheel will activate whatever action it has been programmed to generate in the Control Panel.

So, if the new functionality is desired, one must make sure that the mouse is set up as desired in the Windows Control Panel. In certain cases (depending on the type of mouse) the action of the buttons can be programmed per application, so one could have a particular setting for the CAM software and a different one for other applications.

- **“Setup/Layers” (F10):**

By layer or Board Layer: These two views of this setup dialog are now available. The first: By Layer is the same as the operation of this dialog in previous versions. By Board Layer shows only board layers (one entry per board layer).

When one changes a setting for an entry in this mode, all layers in that board layer will be changed. For instance, changing the color of a board layer will cause the color of all layers in the board layer to be changed.

- **Mouse Right Click**

The context menu (right mouse click) has several new commands. Two commands that previously showed up only when clicking on top of the “B Layer” column heading now appear at all times. The new

commands are

⇒ “Move Up”,

⇒ “Move Down”

The “Move” commands will move (up or down) by one position a sequence of layers starting from the layer that one clicks on and continuing with all successive loaded layers (until the next empty layer is reached). This is useful (for instance) for making an empty spot to insert a VIA layer.

⇒ “Insert **Selection**”

“Insert Selection” will move a selected layer (or range) to the location where one clicks. Loaded layers in the destination range will be moved up to make room for the inserted layers. This command is useful for rearranging the order of layers.

The previously existing commands that now always appear on the context menu are “Renumber board layers” and “Order layers by B Layer”.

- Layers Toolbar:

Context menu (right click). Two menu entries have been added:

⇒ “**Board** Layer ON”

⇒ “**Board** Layer OFF”

These will turn the visibility of a whole board layer ON or OFF that is for all the layers that are part of that board layer.

- Remember Directory for Import

“Setup/Remember Directory for Import”: If set, the File/Import dialog will open in the directory last used for a File/Import operation.

- Import Files

“File/Import” has new features:

⇒ “**Bitmap**”: Reads Windows bitmap files (.bmp) and converts them to polygons. This is useful mostly for generating CAD data for legacy designs (for which only film exists). The first step is to scan the film and convert it to a bitmap file. The bitmap file is then imported into CAMMaster. The ideal input file has one bit per-pixel, where black pixels represent copper and white pixels are the background. CAMMaster will accept other types of bitmap files as well and will attempt to separate copper from background using the color (or gray scale) value of the pixels. The following bit-per-pixel values are supported: 1, 8, 16, 24 and 32.

Ideally, the scanned image should be first pre-processed (using image processing software), to sharpen edges and convert the image to black and white (preferably 1 bit-per-pixel). If that is not possible CAMMaster has two parameters users can set (in the Options dialog):

Treat Dark or Light pixels as copper (basically whether the image is positive or negative).

How to separate Black from White in a grayscale or color image (this only applies to images that have more than 1 bit-per-pixel). The user can specify the intensity of the pixel above which it will be considered white. For RGB images, the intensity is the average intensity of the red, green and blue components.

⇒ “**Drill**”:

added support for the following Sieb & Meyer format commands:

M97, M98 (Drill Plain Text), G84 (Nibble Hole), G85 (Nibble Slot).

- Export Files

“File/Export” has these features:

⇒ “**DPF**”: export in Barco DPF format as documented in “DPF Format Description”, DPF Version 5, November 1998, Barco Graphics.

⇒ “**Drill**”:

For Excellon format: a new option for exporting Step & Repeat blocks: “R Repeat Blocks”. If

checked, repeat codes (R#) will be used when possible. If unchecked, repeat codes will not be used and each M02 block will be output by itself. This is useful for machines that do not support the R codes and to minimize the number of M01 (End of Pattern) commands, as there is a limitation of at most 3 nested M01s (Excellon spec).

For Sieb & Meyer format: support for G84 (Nibble Hole) and G85 (Nibble Slot).

- **Polygons with Holes**

The ability to replace polygons with holes with a set of polygons without holes. This is useful for many reasons. One is that the Gerber syntax for polygons (G36/G37) does not allow specification of certain polygons as being holes inside other (containing) polygons. As a result, in order to represent polygons with holes in Gerber one has two options:

Make the holes scratch entities (on a separate layer)

Decompose the polygons with holes into a set of polygons without holes (which fit together similar to a jigsaw puzzle).

Method (2) has been added in this release. It appears in two places:

- ⇒ **“Edit/Edit Selection/Polygons/Decompose Into Strips”:**

Will replace any selected polygons that have holes with a number of horizontal polygonal strips without holes, such that the result will represent the exact same copper (same image).

- ⇒ **New Tab:**

A new tab in the “File/Export/Gerber” Options dialog called “Polygons”:

It has two choices for exporting polygons with holes: “Put holes in scratch layer” and “Decompose into strips”. Note that this does not modify the database itself, it just controls how the polygons are exported. Also note that this tab does not appear for exporting MDA Gerber data, as polygons with holes can be represented using POIN/POEX D Codes.

- **Custom Shapes - Polygons with Arcs**

(D Codes of type M) can now contain polygons with arc edges. Previously only polygons with straight edges were stored and arcs were replaced with approximating straight edges.

These types of shapes can be generated from several sources:

- ⇒ **File import**

- ⇒ **D Codes table/Dimensions/Build Shape:**

when the frame includes polygons with arcs

- ⇒ **Tools/Convert Drawn Pads/Global:**

when the convert pads included polygons with arcs Note that if these shapes exist in the data and a binfile is saved, a new version of the binfile will be written (version 7) which will not be accepted by older versions of the software.

For exporting Gerber extended variants: The standard Gerber extended does not have a polygon primitive with arc edges (for custom shapes). When exported these edges will be replaced with chords. The MDA extended Gerber format allows arc edges for the polygon primitives.

- **New Edit Selection Features**

“Edit/Edit Selection” has these new features:

- ⇒ **“Swell”:**

a new option has been added - “Transcode all swelled D Codes”. If checked will force all D Codes that are swelled to be transcoded even when no unselected elements of that D Code exist. This is useful when preserving the integrity of the original D Codes is desired.

- ⇒ **“Match Two Points”:**

This is a general purpose transformation which combines translation (move), rotation and scaling in one step. The user picks two pairs of points. The transformation is done so that the first point in each pair is transformed into the second point.

Note that although all elements in the selection are transformed, the D Codes with which they are drawn are not. Because of this, it is most useful for transforming polygons.

⇒ **“Step And Repeat”:**

added the ability to edit and change step-and-repeated data. The procedure is the following:

(1) Select the step-and-repeated data.

(2) Do “Edit/Edit Selection/Step And Repeat/One-up”. This will remove the step and repeat from the selected data and make the result ordinary data that can be edited and modified. Up to this point all of the above was available in previous versions. What we have added is that after “One-up” is done the “Step And Repeat” menu will show the step-and-repeat parameters that have just been removed (counts and displacements) at the bottom of the menu (in a list of up to 5 recent step and repeat parameters). These parameters can later be reapplied by just selecting the menu item.

(3) Edit the one-up data as desired.

(4) Select the data for which the step-and-repeat needs to be restored, then pick the step-and-repeat parameters from the bottom of the “Step And Repeat” menu.

When step and repeat blocks are nested, this procedure may need to be repeated several times if the data to be edited is in some inner step-and-repeat block.

⇒ **“Polygons”**

“Separate Holes”:

For all polygons with holes in the selection: remove the holes and add as separate polygons.

“Group Holes”:

Group all the contours in the selected polygons and attach as holes contours that are included in others. A contour is defined as a closed path component of a polygon and could be a container or a hole.

Note that for polygons that do not overlap a “Separate Holes” followed by a “Group Holes” will reconstruct the original polygons.

“Replace With Fill”:

Selected polygons will be replaced with filled areas using traces of a D Code specified by the user. This D Code must be of shape “C”. Note that to achieve the same result in the past, the polygons had to be converted to F outlines and then one could use “Tools/Fill Polygons/Solid”.

- **Insert Text Spacing**

Extra Spacing - items added for extra character and line spacing. Character and line spacing can be increased by entering the desired values in the dialog boxes. Use 0 for default spacing.

- **“Go”:**

⇒ **“Center Of Selection”:**

moves the cursor to the center of the bounding box of the currently selected elements.

⇒ **Center of Element:**

“Active Element/Center of Closest”. Similar to “Active Element/Closest” except that the cursor is moved to the center of the element. Most useful for polygons. Shortcut is Ctrl+Shift+A.

- **Shortcut keys:**

⇒ **Layers greater than 9:**

To make active a layer number larger than 9. It was previously not possible to quickly set active a layer number that had more than one digit. Pressing any of the digit keys would make the corresponding layer active.

To set higher numbered layers the Ctrl+Shift modifier has been added. So to set the active layer to 24 and should do the following:

Press Ctrl+Shift, while keeping Ctrl+Shift pressed press 2, then 4, then release Ctr+Shift. This would be 3 keystrokes.

Note that this does not save many keystrokes over the already existing way (“Setup/Layer/Active (L)”), which would have required:

L, 2, 4, Enter (4 keystrokes).

However, some people seem to prefer the new method.

⇒ **Dimensioning Tool:**

For the dimensioning tool (Insert/Dimension): Ctrl overrides the “Extended bars” setting in the setup dialog.

- View Loaded Data From 0,0

“View/Zoom/...Include (0,0)” This affects the operation of the “View/Zoom/To Show All” (Home) command. If “...Include (0,0)” is set then the origin (coordinates X=0, Y=0) is always included in the data displayed even when it is outside the loaded data (this is how Home command worked until now). Sometimes it is desirable to only display the loaded data, as the origin can be far away from it. The user has now the option to pick the preferred mode of operation for this command.

- New Contour DRC Features

“Tools/Design Rules Check/**Run Contour DRC**”:

⇒ “Check Gaps with Same Nets”: option **has** been added.

⇒ “**Soldermask Gap**”:

“Mask to mask”, new rule added. This will check the pad to pad air gap in the soldermask layer (when set to a non-zero value – when set to zero this rule is not used). Previously the check was done by using the “Trace to mask” rule.

- New Netlist Features:

“Tools/**Netlist**” now has the following new features:

⇒ Extract **Contour** and Extract Raster: Blind vias are output as 317 records

⇒ Extract **Contour**: If a through hole has no round covering pad, one will be automatically generated for it by picking the largest noncircular SMD (with same coordinate) as its covering pad. The SMD would still be output in the netlist

- Changes to Printing Routines

“File/Print”: The graphics rendering routines for printing have been changed. Windows GDI routines are now used instead of rendering to a bitmap and then sending the bitmap to the printer (as was done previously). As a result, printing should now work for all printers. Previously some printers did not work properly because of memory limitations (the bitmaps were too large) or raster operation limitations. This change should also allow capturing the image in a PDF file (with Adobe Acrobat Distiller) or a PostScript file.

Note: This new method does not work on Windows 98 or ME, in which case the old method (bitmap based) will still be used. It does work on Windows NT, 2000, XP, 2003 Server.

The color output options have been reduced to two: “black on white” and “colors on white”. Note that colors will not be translucent when printed (this is different from how the colors show on the computer screen). When running under Win98 or ME an additional check-box is shown called (“Negative”) which will cause the bits in a “black on white” output to be inverted (this is needed on some printers).

- “**File/Import/Job/Options**”

Origin selection has been removed. Origin is now always at (0,0). Allowing to set the import origin in a sub-dialog was confusing and dangerous. The imported data can always be moved later if that is desired.

- “**File/Import/Gerber**”:

The user is warned if a self-intersecting polygon is encountered in the input.

The “Full circles in polygons” settings has been removed from “Options/File Interpretation”. The “Ignore” choice caused full circles to be silently ignored, which was a problem for many users.

- **“File/Import/Netlist”**
Misaligned X and Y coordinates in the “Feature Location” field (columns 42-57) are now accepted. The IPC spec states that they should be right justified, but we have seen files where they are left justified. Previously this was considered an error.
- **“File/Import/Directory/Zip”**
Files that contain all the necessary format information such as Extended Gerber will be displayed in “blue”. The options menu now has a check-box “Format unknown files to formatted” If selected, files that do not contain all of the format information (such as drill files) will be formatted/scaled to fully formatted files (such as Extended Gerber). These files will be displayed in green and can still be edited.
- **Aperture Wizard**
A new check-box has been added in the third menu titled “Treat commas as decimal ex: 1,00 = 1.00” This will allow the aperture wizard to import files using the European style of decimal identifiers.
- **“File/Export”:**
 - ⇒ **Gerber Extended:**
Custom apertures that contain arcs cannot be exported in Extended Gerber because there is no corresponding command to represent them. Previously this caused error 115 and the export aborted. The user then could have converted the custom aperture to replace the arcs. In this version arcs in custom apertures will be automatically exported as sequences of chords.
 - ⇒ **Gerber Varieties:**
Improved rounding of polygonal data when exporting to lower resolutions (lower than .6 in English or .5 in metric).
- **Setup Layer Improvements:**
“Setup/Layers” (F10): Position and size of the dialog as well as the widths of columns in the table are now saved in the registry and restored next time the program is started. Note that the width of the columns can be modified by the user by dragging the column separators.
- **Replicate to Non-Contiguous Layers**
In the Layers Toolbar; layers can now be replicated to a number of layers that need not be contiguous. Previously only a range of contiguous layers was allowed as the destination.
- **“Edit”:**
 - ⇒ **Chords to Arcs:**
“Edit Selection/Polygons/**Chords To Arcs**”; improved detection of arcs.
 - ⇒ **Transcoding:**
“Edit Selection/**Transcode**”; The “First Unused” option has been changed to “First Unmapped”. This will pick the first D Code that is either not present in the D Code table or has shape of type “?” (unspecified).
 - ⇒ **Aligning Layers:**
“Move Layers/**Align**”; the guide element needs to be selected and to be the only selected element. Previously the guide element was the one from the last “element align with properties” (Shift+A) operation. The new method makes the identity of the guide element clearer. Also, previously the guide element was only matched against elements of the same D Code. There is now a new setup command “Move Layers/Align Method” that allows the user to pick the method to be used: same D Code (as before), same shape type and same shape (shape type and shape dimensions).

- **TEXT is now Grouped**

The internal representation of text has been changed.

In previous versions text was stored as a sequence of strokes (traces or pads) which were not distinguishable from other (non-text) traces and pads. With this new implementation text is a separate entity (a composite element) so that it can be modified later (by editing the text string, changing the font, rotation etc.) This modification can be done with “Edit/Edit Selection/Text/Change” after selecting one (or more) text blocks. When selecting only one text block, all parameters of the text (including the text string) can be changed. When more than one text block is selected, only parameters that have the same value can be changed (the others are grayed out). An exception to this rule is the font, which can be changed even if the selected blocks have different fonts.

Note that there are other methods to change parameters that are not common to all blocks. For instance one can change the D Code of a group of text blocks by using “Edit/Edit Selection/Transcode” (as one would for other elements).

One can ungroup text by using “Edit/Edit Selection/Text/Ungroup”. This will transform the selected text blocks into their stroke representation (normal traces or pads).

- **Select Text:**

The Selection Toolbar criterion dropdown has a new added item called “Text” which can be used to select or navigate over text blocks. Text blocks can only be edited with operations that would result in another text block. So, for instance, moving, rotating and mirroring are allowed, whereas scaling can preserve text blocks only if the two scale factors (X and Y) are the same. If there are not the same, scaling will be done but the text will be ungrouped before the scaling operation.

For other commands, if one finds that they do not affect text blocks in the desired fashion the work-around is to ungroup the text blocks first.

Text blocks will be ignored by most of the commands under “Tools”, except Netlist, DRC and “Generate Soldermask”. Note that for netlist extraction the copper of text blocks is taken into consideration when computing connectivity (of other circuit elements), but no net numbers are ever assigned to text blocks. Binfiles that contain blocked text will not be readable by older versions of the CAM software. If you need to send such data to users of older versions make sure to unblock all text blocks before saving the binfile.

- **“Insert/Loc and Dim Setup”**

a new parameter has been added: “Max digits to the right of decimal”. This can be used to control the precision of the inserted text for location or dimension. Possible values are from 0 to 6 inclusive.

- **“Tools/Design Rules Check/Load Errors from File ”**

More than one file can be loaded at the same time (with the same load operation). This allows the display of the contents of several files simultaneously. One example where this would be useful is for viewing together the results of several different DRC operations done with per layer rules.

- **“Tools/Netlist/Extract Contour”:**

- ⇒ Improved handling of paint and scratch layers
- ⇒ The method for determining covering pads has been improved
- ⇒ Pads with holes are not output
- ⇒ Lloyd Doyle layer-by-layer export: N/C drills are not output for each layer (other than drill layer)
- ⇒ Thru holes with no round covering pad will have one automatically generated by picking the largest noncircular SMD. The SMD will be output in the netlist. When the file is imported into Probe or Fix, the user can choose to test for hole or pad (Setup/Configuration/ Operations/ Remove Padstack - Keep Hole - Keep SMD)

- **Netlist, Extract Contour and Extract Raster**

- ⇒ Endpoints identification has been improved

⇒ Blind vias – are now outputted as a record 317 (thru hole) with the relevant access code set as well as drill and covering pad sizes

- **“Convert Drawn Pads/Global”**

The algorithm for detecting and converting the pads has been improved

8.2.21

- Auto-Snap to Elements:

“Go/Closest/Auto-Snap”. When activated, will cause the cursor to snap to the closest active element before certain operations: element insertion, move etc. These are the same operations that cause the cursor to snap to the closest grid point when the grid is on. Auto-Snap uses the same criteria as Go/Closest/Align (“A”) except that a tolerance is in effect: elements that are farther than the tolerance will not be considered.

When both the grid is on and Auto-Snap is active, Auto-Snap is tried first.

One example when this is useful is when inserting traces that need to align with existing traces or pads.

- Dimension Tool:

“Insert/Dimension”

New dimensioning tool (similar to the “Insert/Location” tool).

Will insert dimensioning text and arrows. It is a three-step operation:

(1) Pick the point to measure from

(2) Pick the point to measure to

(3) Pick the desired location of the dimensioning text, arrows and bars

There is a separate setup dialog (Insert/Loc and Dim Setup) for specifying the text size and font as well as various options. This dialog is common to this and the Insert/Location tools. It can also be activated using a context menu (right click) while any of these two tools is active. The options that are specific to the dimensioning tool are:

- ⇒ **Forcing Measurements**

to always be horizontal or vertical (depending on whether the vector from step (1) to step (2) is closer to the horizontal or vertical). Note that this setting can be temporarily overridden by pressing the Shift key while picking point (3).

- ⇒ **Extending the Bars**

to reach close to the step (1) and (2) points.

- Polygon Features

“Edit/Edit Selection/Polygons”

- ⇒ **“Make Well Formed”:**

Replaces any malformed polygons in the selection with well formed ones. Well formed polygons have no intersecting edges (except at vertices) and no overlapping edges.

- ⇒ **“Chords To Arcs”:**

Approximates sequences of chords with arcs for polygons in the selection. The user can specify a tolerance. The new selection will consist of the changed polygons. If none were changed, the selection remains as before.

- ⇒ **“Arcs To Chords”:**

Replaces arcs with sequences of chords that approximate them. The resulting polygons will contain no arcs. The new selection will consist of the changed polygons. If none were changed, the selection remains as before.

- Convert Drawn Pads to Flash

“Tools/Convert Drawn Pads/Global”

This is a new way of converting drawn pads to flashed pads that uses contouring methods and acts on

selected elements. Drawn pads in the selection will be replaced with flashed pads and the resulting selection will be the newly generated flashed pads. There are two methods available: “standalone drawn pads” and “using a guide layer”.

The standalone pads method should be used when the drawn pads are not continued with traces (or when such traces have been removed from the selection).

The guide layer method uses elements in a different layer (the guide layer) to pinpoint the location of the desired drawn pads. Only those elements in the selection that are completely covered by drawn areas in the guide layer will be used for finding drawn pads. For the purposes of this computation the drawn areas in the guide layer can be inflated or deflated by a “Margin”, if desired. Usually the guide layer is the soldermask layer, but it could be any other layer chosen by the user.

Once blobs have been isolated (in either of the two methods) only those whose max dimension is within the user specified min and max (“From” and “To”) will be considered. The resulting contours are checked against the following standard shapes: C, S, R, O, D, E and Q using a user specified “Tolerance”. If none of these standard shapes match and the “Generate custom shapes” check-mark is set, then M type D Codes will be generated for these shapes. There is a sub option to this for allowing for concave custom shapes. Normally such shapes are undesirable as flashed pads. An example of a concave shape is a cross, whereas an octagon is a convex shape. Holes are only allowed in C and S shapes. Note that M shapes will contain a polygonal outline. Since the extended Gerber specification does not allow for arcs in custom shape outlines, all arcs will be converted to approximating chords.

The elements in the selection that have been matched will be deleted and replaced with flashed pads.

Note that if selection already has a pad that covers the whole area at a conversion spot, that pad will be preserved (existing D Code preserved) and any elements covered by it will be deleted.

The starting D Code for the newly generated pads can be specified by the user, as well as the resolution of such shapes (all newly generated shapes will be rounded to the specified resolution).

- **Select by Intersection**

“Select/By Intersection”. Narrows an existing selection depending on the intersection of elements in the selection. The selected elements in the reference layer will be checked against the selected elements in the destination layer. Only those elements in the destination layer selection that satisfy the intersection criteria are preserved in the selection (all other elements are removed from the selection, even from other layers). For the purposes of computing the intersection, the elements in the destination layer will be considered inflated/deflated by a given margin (which can be 0).

The destination layer and the reference layer can be the same in which case the selection will be tested against itself (all elements in the selection versus all others).

Elements that have zero width either as they are in the database or after being deflated by the margin will be ignored during this operation.

- **DRC Enhancements**

“Tools/Design Rules Check” now has these new improvements:

- ⇒ **Polygons and Contour DRC:**

“Run Contour DRC” is now handling polygons. Previously polygons were ignored. To control what rules apply to polygons there is a new setting in the “Options” tab called “Treat Polygons As” with two possibilities: “Traces” and “Pads”.

- ⇒ **Gaps within Same Net:**

New in “Options” tab: “Check gaps within same net”. When ON, gap errors will be checked even for elements that are in the same net (electrically connected).

- ⇒ **Display Errors with ‘X’:**

“Display Errors” has an added option “X marker for all”. When ON, an X marker will be superimposed over the display of all errors. This is to facilitate finding errors for which the display vector is short. These are hard to see unless one is zoomed in on them. Previously the X marker was only displayed for errors that had a vector of 0 length.

- ⇒ **Saving Errors to a File:**

Commands to save/restore the error information to/from a text file: "Save Errors To File", "Load Errors From File".

This is useful as a way to communicate this info to coworkers or save the information for future examination. Previously the DRC error info was lost when the job was closed. In addition, the resulting files are text files and could conceivably be used for further processing by custom software.

- "Edit/Insert":
 - ⇒ **Menu Location Has Moved:**
The position of the menu has been moved to "Insert" (top-level menu), because this is a very commonly used submenu. Note that the shortcut key (Insert) still works even with the new location.
 - ⇒ **Insert:**
When inserting a trace-type (linear or arc), the outline of the shape to be inserted will be displayed around the cursor as a guide to the user.
- Setup Selected Tools:
A new group has been added: Setup/Tool Codes (Shift+F5) with two buttons:
 - ⇒ **"Type + Diameter"**
Same as double clicking on the selected tools and will set the type and diameter of the selected tools.
 - ⇒ **"Type Only"**
Sets only the type of the selected tools, keeping their diameters.
- Malformed Polygons in Contour Operations
All contour-based operations: contour netlist, DRC etc. will now properly handle malformed polygons (see "New Features" for definition). Previously the operations would abort when such polygons (self-intersecting) were found.
- Extended Gerber Parsing
Changed to where LP (layer polarity) commands do not reset pending G36 (polygon area fill) commands. We have seen customer files of the form:
`G36*%LPD*%X-1757Y13D02*...G37*`
Previously the LPD command would reset the G36 and the polygon would be lost (replaced with outline traces).
- Compare Layers Supports Polygons
"Tools/Compare Layers": support for polygons has been added. Previously polygons were ignored during this operation. Also the option to transfer to another layer has been removed and the results are always returned as the new selection. If so desired, the user can always transfer the result after the comparison is done.
- Transcoding D Codes
"Edit/Edit Selection/Transcode" (Ctrl+T): When the user is prompted for the D Code to transcode to, a choice is given between the current D Code and the first unused. Previously only the first unused would appear. Some users preferred the current D Code as they have just set it up in the D Code table in preparation for the Transcode operation.

8.1

- Save File (Ctrl+S)
Normal Windows shortcut for File/Save (Ctrl+S) has finally been implemented. We had postponed the

implementation of this shortcut because Ctrl+S used to have a different meaning in our DOS products.

- **BMP File Export**

“File/Print to .bmp File”: a way to generate standard Windows bitmap files from the currently loaded data; either the current view or what is inside the frame. The image can be combined layers or one board layer at a time (with one .bmp file per board layer). Besides other uses, this can be another way of printing in cases where direct printing (to printer) does not produce the desired results.

- **File Import Enhancements**

“File/Import” now has the following:

- ⇒ **Inexact Arcs:**

“Gerber/Options”: added an option in the “File Interpretation” tab called “Inexact arcs”. This controls how inexact arcs (arcs that have different radii at the two endpoints) are interpreted. One can pick one of two ways to deal with these arcs (as they are illegal as presented):

- ⇒ **“Move center”**

If selected then the center of the arc will be moved to make the two radii the same. This is sometimes dangerous as the guessing for the arc orientation can be wrong resulting in “Mickey Mouse ears” (large multi-quadrant arcs where short single-quadrant arcs were intended by the user).

- ⇒ **“Add connector”**

If selected then the center is preserved and one of the arc endpoints is moved to be closer to the center. A connecting linear “bridge” is added to go to the original endpoint (so that electrical connectivity is not lost). This would cause an arc to be replaced by an arc and a (short) linear trace.

- ⇒ **Ignore G92/93:**

Added an option called “Ignore Excellon G92/G93” in “Drill/Options”. When set causes G92 and G93 commands to be ignored. These are “Zero set” commands that change the origin.

- **DRC Violation Colors**

DRC violation errors display colors are now saved in the .cfg file.

- **Net Compare Enhancements**

“Tools/Netlist/Netcompare/Compare Two Netlists”:

- ⇒ **Mismatched N/Cs:**

A new option has been added: “Mismatched N/Cs”. If checked will flag as errors nets in REF and NUT that are N/C on one side but not N/C on the other. In previous versions N/Cs in NUT that were matching a non-N/C in REF were flagged as “NC Open” and there was no way to disable this behavior. Now this will be flagged only when the above option is checked. In addition (when the option is checked) nets that are N/C in REF but non-N/C in NUT will be flagged as a new error type: “NC short”.

- ⇒ **Inconsistencies:**

The REF netlist can be checked for inconsistencies before processing starts (turned on or off via a newly added check-box).

- **Define Report Filename:**

“File/Report/Set Name”: to set the report file name. The report file is used in many operations where the name cannot be set, such as for generating D Code transcoding reports while importing Gerber files.

- **Screen Toolbar Display Boxes Widened**

Widened X and Y coordinate display boxes in Screen Toolbar to allow the display of all the digits.

- **Individual User's Preferences**
The product configuration files (*.cfg, for instance CAMMaster.cfg) are now stored in a different directory to facilitate per user configurations. They used to be stored in "Windows directory" (normally C:\Windows). Now, if the environment variable "HOMEPATH" is defined, that directory will be used instead. This variable is automatically set in certain versions of Windows, such as XP, but can be set by the user otherwise.
If you want to preserve you're your old settings, please move the .cfg file to the HOMEPATH directory. To determine the setting of HOMEPATH or to set it to a different directory, use "set" from the command prompt.
- **Move to Cursor has Independent Relative/Absolute Mode**
In the "Move Cursor To" dialog, the type of move (Absolute, Relative, Incremental) is now set separately depending on the coordinate display mode (Absolute, Relative), that is once set in Absolute display mode it will stay the same even if changed while the display mode is Relative. The two settings will be remembered independently.
- **Report Files List Both Import and Export Formats**
The report file will now separately list the Gerber import and export formats, as well as the Drill import and export formats. Previously only the import formats were reported.
- **Custom Apertures with Polygons**
"Setup/D Codes" (F5): "Custom Apertures/Replace With Primitives": primitives that are polygons will now generate polygons rather than F-type outlines.
- **A.O.I. Export**
A.O.I. International and Lloyd Doyle netlist export is now standard in CAMMaster.

8.0.X

- **Contour DRC now Available:**
(Tools/Design Rules Check/Run Contour DRC). This method has the advantage over Raster DRC that it is accurate and resolution independent.
- **Import Gerber via File/Open:**
One can now import Gerber files using the File/Open dialog. This has the additional benefit that one can associate a common Gerber file extension (such as .ger) with these executables, so that double-clicking on that file would launch the executable. The .bin extension is automatically associated with the PentaLogix CAM products (excluding ViewMate).
To create the association: right click on the desired file and select "Open With", then choose the executable. One more step is needed to make sure that path names that have spaces in them are handled correctly. Open Explorer, then "View/Folder Options/File Types".
Pick the type you just added and click on Edit, then "open", "Edit". In the "Application used to perform action", add "%1" after the name of the executable. The double quotes around %1 are needed to make sure that the entire path name is passed to the executable.
- **"File/Import Enhancements" (Gerber)**
 - ⇒ **Mirror Image:**
Support for "MI" (mirror image) RS-274X parameter.
 - ⇒ **Padstacking:**
A new option in "Options/File Interpretation". "Consecutive flashes at same position". This has two options: "Keep only one" and "Keep all". The previous versions would always "Keep only one", but apparently some users need these padstacks preserved.

- Drill Import Options
“Setup/Layers” (F10): added a button to bring up the drill import options.
- Messages Mode – Background
“Setup/Messages Mode”: new state added “Background”. This is the “quietest” one (more so than “Batch”). When in this mode, error messages will be displayed in the status line rather than via a dialog box and as such will require less user interaction. Also the graphics view is not refreshed in this mode, layers toolbars are not updated etc. This is only useful for CAMMaster scripting.
- Goto Favorite Position
“Go/Favorite Positions”: a quick way to go to a favorite (X, Y, Zoom) triplet that one can previously record.
- Gerber IPNEG Followed by IPD or IPC
For both import and export of Gerber Extended files: a new option in the “RS274-X Options” tab. The option is “Negative layer is IPNEG followed by:” and the two possibilities are “LPD” and “LPC”. It appears that various plotters and CAM packages expect different commands to render a negative layer: either “IPNEG” followed by “IPD” or “IPNEG” followed by “IPC”. As usual, the Gerber manual is no help, as it does not elaborate as to what the correct combination should be, so we decided to let the user pick one or the other (separate for import and for export). Previously “IPNEG” followed by “IPD” was the only interpretation.
- MDA Output – Unique Layer Names
“File/Export/MDA Autoplot/Options/MDA Options”: a new check-box called “Force unique layer names”. This is only relevant when “Merge paint and scratch” is also checked and will append “.L<layer_number>” to each layer name to insure that the individual files generated have distinct names.
- Select Invalid Polygons
A new selection criterion in the “Special selection” droplist called “Invalid Polygons”. This will select polygons that are invalid as far as the PentaLogix CAM software is concerned (for instance self-intersecting polygons are invalid). The intention is to replace this command in the future with one that can “fix” the invalid polygons.
- Handling of negative and scratch layers
Layers that were set as both negative and scratch (NS) were handled differently in the contour methods versus the raster methods. A unified way of handling them has been implemented following the rules below:
 - ⇒ **A scratch layer cannot be the first layer in a board layer.**
Such a situation will be flagged as an error during processing of connectivity information. We have sometimes seen the first layer set to both negative and scratch. If the user’s intention was to have it as a paint layer then both the scratch and negative settings should be removed. If the user’s intention was to have it as negative, then the scratch setting should be removed.
 - ⇒ **If an NS layer follows a negative layer**
(N = Negative set, but not Scratch), then the NS elements will be scratched from the N elements and then the whole image will be inverted (to make it negative)
 - ⇒ **If an NS layer follows a non-negative layer**
(P=Negative not set), then the NS layer will be first inverted (to make it Negative) and then the result will scratch P. This will make an NS following a P act as scratching everything except where there are elements of NS.
This will affect Raster and Contour netlist as well as Raster and Contour DRC as well as AOI tools.

- Layers Toolbar is Longer
The layers toolbar is now longer.
- Floating License Manager
The floating license manager now correctly administers number of seats allowed.
- Loading 10,000 D-Codes
All possible D Codes (10000) can be loaded at the same time (previously there was a limit of 2000 simultaneously loaded D Codes).
- 255 Layers
The number of layers has been increased from 99 to 255.
- Save in High Resolution
“File/Save Options/High Resolution” is now set by default. Previously the setting was remembered between software activations.
- Faster Contour Netlist Extraction
This was done by algorithmic improvements as well as special handling for crosshatched plane layers (these used to take very long to process in previous versions). To take advantage of this new feature, the user needs to mark layers that have crosshatch plane patterns as “Fill” (check-mark present) in the “Setup/Layers” (F10) dialog.
- Negative and Scratch Rendering
The rendering (display on screen) of negative and scratch layers has been improved. For the case of negative layers: when more than one negative layer existed the rendering was sometimes incorrect in the past, since it did just one negative operation on the merged negative layers. For scratch layers: they will automatically be displayed in the color of the immediately preceding paint layer, so that manually setting the layer color to match is no longer necessary.
- License String Improved
License string installation is now done with an external executable (InstLicense.exe) and the pesky “0001.tmp” files are deleted when application closes.
- G75 and G74 Gerber Export
When exporting Gerber files, G75 or G74 is output only once per layer.
- “File/Import”:
 - ⇒ “HPGL”:
“PG” command (“Advance Full Page”) will cause step to next layer. Previously it was ignored.
 - ⇒ Excellon:
For **Excellon** drill files; user overridden format settings (m.n, metric, English, etc) will be preserved. Previously they reverted to the settings in the drill file itself.
 - ⇒ ODB++
File import of ODB++ is now supported.
- Distant Measurement Handles All Types
“View/Distance/From Edge”: distance measurement tool from edges of elements now handles all element types. Previously it only handled linear traces and certain pads (of simple shapes).

- Sum of all trace lengths
“View/Selection/Properties”: a new item has been added – “Sum of all trace lengths”. This will appear when traces are part of the selection. When nets are made of chains of head-to-head traces, this will also be a good approximation for the length of the part of the net that is selected.
- “Setup/Layers” (F10):
 - ⇒ **Paths:** The “Path” button has been expanded to give a choice of 3 operations:
 - Strip path
 - Make relative to current directory
 - Fill in full path.
- Viewing Text Files
The application used to view text files (such as by File/Report/View) used to be “notepad.exe”. It has now been changed to use any application that the user registered for opening a “Text Document” (.txt extension). This is usually done with “Explorer/View/Folder Options/File Types”.

11.24.55

- **EMS Magician**

MouseBite must validate the position to not crash to another MouseBite or edge segment
Rails were not using Advanced Rail Option values
Add fiducials in rails is not working
Rails should be applied in all the panel

11.24.43

- **Import ODB++**

Failed when importing .zip files

- **EMS Magician**

The option to insert mouse bites using 'Closest to click location' is not working at all.
Panelizer program is generating wrong mouse bites
Removing mouse bites and inserting them again is working wrong.
Sometimes mouse bites are added to the lower-left corner instead of the selected position.
Adding 2, 4, or 5 mouse bites doesn't display the holes correctly.
Finalizing the process to add Mouse Bites is not generating the array.

11.24.40

- **DFM**

Some Feature Size violations were missed

- **Licensing**

Failed when the license file (.lic) did not have the standard name

11.24.20

- **Export Drill**

The “F, S, B, H, Z parameters, if present” and “F, S, R, N parameters, if present” did not work as expected.

- **Import DXF**

11.24.11

- **D Codes Table (F5)**

Clicking on Angles column header to sort did not work as expected.

- **Stencil Generation**

Bugs fixed.

- **DFM > Spacing Violations**

Crashed sometimes when “Cancel” was pressed.

11.24.3

- **Export Gerber X2**
Crashed on certain data.

11.24.2

- **Arcs**
Converting arcs between English and metric units generated incorrect arcs in some cases.

11.22.50

- **“File > Open” did not update the job name.**

11.22.44

- **Convert Negative to Positive Crash**

11.22.41

- **Tools > Convert Drawn Pads > Global**
Slightly different shapes were recognized as the same.

11.22.37

- **Insert Text**
The “Group” option did not work.

11.22.36

- **Import ODB++ bug**

11.22.30

- **Polygon Merging**
- **Tango Aperture Import**
- **Export DPF**
Incorrect for rotated shapes

11.22.23

- **Incorrect rendering of small polygons at certain zoom levels**

11.22.21

- **Convert Layer to Positive**
Used the frame instead of board outline layer.

11.22.16

- **Import ODB++**
s_ths import problem.
- **Polygon Merging**
Took a long time when scratch polygons involved.

11.22.12

- **Component Layers**
Some mirrored and rotated components were displayed incorrectly.
- **Gerber Export or Rotated Rectangles**
Rotation was lost in certain cases.

11.22.10

- **Stencil Homebase**
The facing out option was ignored.

11.22.2

- **D Code mirror**
Failed on rotated D Codes.

11.20.25

- **Gerber X2 import**
TO and TD directives misinterpreted.

11.20.22

- **Import > Directory**
Failed on certain data

11.20.19

- **Bug in Paint and Scratch > Optimize**
- **DFM > Feature Size Violations**
Some bugs fixed.

11.20.16

- **Import ODB++**
Some step-and-repeated ODB files were imported incorrectly.

11.20.12

- **Tools > Stencil Apertures**
V Homebase did not work.

11.20.11

- **Feature Size Violations**
Reported incorrect results.

11.24.43

ZoomOnFramePreserveCursor
ZoomShowAllPreserveCursor
SelectClosestExPreserveCursor

11.24.30

RemoveUnusedPackages
SetSelectionToolState

11.24.23

ExportSTL

11.24.20

PocketRoutSelection

11.24.15

ChangeComponentRefDes
ConvertFootprintsToComponents
ConvertOneFootprintToComponent
GetAllComponentsInfo
SelectComponentByRefDes

11.22.48

SelectElementsWithAreaSmallerThan

11.22.30

RenumberBoardLayers

11.22.25

ConvertDCodesToCustom

11.22.23

LayerOr, LayersAnd, LayersMinus

11.22.12

DcodeShapeAndAngle

11.22.1

GetLayerElementCountsEx
GetLayersWithData
RestoreParametersFromFile

11.18.46

GetSelectedElementsCenters

11.18.44

ImportDXFUnits

11.16.34

NanometerResolution
OffsetSelectedPolygons
MirrorSelectedEx
RotateSelectedEx

11.16.12

ImportGDS_TXT
ExportGDS_TXT
DisableApplication
CombineShortStraights

11.16.7

SelectEx

11.14.59

Undo

11.14.35

SetLayerColorsFromLayerType
LoadLayerTypeToColorMap
CreatePackageFromCAMData
CreateCAMDataForPackage
ImportPackages
ExportPackagesPentalogix
ExportPackagesODB
ImportComponents
ExportComponents

11.14.10

PrintToSVG

11.12.54

About

11.12.9

ConvertPadsToHoles
SortDrillHoles

11.10.79

SmallestCoveringRectangle

11.10.69

SplitSelectedRoutingPathAtLength

11.10.44

AlignToActiveElement

11.10.43

InsertThreePointArc
PolySkeletonBrushWidth
PolySkeletonMethod

11.10.40

BreakIntersectingTraces
PolygonsToSkeleton
SelectChainEx
SelectedRoutePathLength

11.10.21

SetRoutePlungeClosestToPoint

11.8.24

ViewCenter
DFM_Drill

11.8.1

ImportDXFOVERRIDEUnits
SelectionDisplayMode

11.6.35

SelectTouching

11.6.26

AreaAndCircumferenceOfSelection
StencilDCodeNumber
StencilComponent

11.6.18

StencilWindows
StencilHomebase
StencilInvertedHomebase
StencilFromModel
GetWindowHandle

11.4.0

LastItemCount

11.2.58

CombineLayersByBoardNumber
TransferLoadedLayersRange

11.2.47

RoutingUngroupToTracesEx

11.2.35

ClipToPoly

11.2.22

SelectClosestEx

11.0.80

BoardRoutIsOK
CancelUserOperation
CSVSeparator
GetInfoForSelectedPolygons
GetSelectionToolNumbers
RoutingAutoLead
RoutingSort
SelectElementsOfToolNumber
SetFocus

11.0.74

RoutingChamferRadius
RoutingChamfers

11.0.71

- New:
 - ⇒ CenterSelectedToFrame
 - ⇒ InsertFrameSizedContour
 - ⇒ InsertFrameSizedPolygon
 - ⇒ MToCSelected

11.0.42:

- New:
 - ⇒ ImportGuessEx
 - ⇒ IsBusy

11.0.9:

- New:
 - ⇒ ExportRoutOptions

11.0.1:

- New:
 - ⇒ AutoAlignDrillLayer
 - ⇒ ExportGBXOptions
 - ⇒ SelectIntersectingTraces

10.6.50:

- New:
 - ⇒ IntersectLayers

10.6.34:

- New:
 - ⇒ DeleteDCodesIfUnused

10.6.0:

- New:
 - ⇒ Color
 - ⇒ ColorsSetup
 - ⇒ CompactLayers
 - ⇒ SetupLayerColorDialog
 - ⇒ WellFormedMethod

10.4.30:

- New:
 - ⇒ DrillPathLengthInLayer
 - ⇒ SaveJobData
 - ⇒ ToolCount

10.2.45:

- New:
 - ⇒ ConvertNegativeToPositive
 - ⇒ DFM_FindMinimumSpacing
 - ⇒ DFM_FindMinimumWidth
 - ⇒ DFM_SpacingViolations
 - ⇒ DFM_WidthViolations
 - ⇒ GetCopperAreaValues
 - ⇒ RoundDCodeSizes

10.2.18:

- Changed:
 - ⇒ FtoPolygons

10.2.0:

- New:
 - ⇒ ImportGuess
 - ⇒ GetLockingCode
 - ⇒ GetStepAndRepeatIDs
 - ⇒ LayersForInsert
 - ⇒ SelectionCrossing
 - ⇒ SelectStepAndRepeat
- Changed:
 - ⇒ PentaLogix geometry files are now at version V2. The only change is that the arc representation includes the center of the arc also (as an information only item, since the arc is uniquely determined by the other three points given). So the syntax for an arc is now:

P1, P2, P3, (Center)

Where P1 and P3 are the arc extremes, P2 is a point on the arc (preferably the midpoint) and Center is the center of the arc (for information only).

10.0.10:

- **New:**
⇒ ToolNumbersTranscode

10.0.0:

- **Changed:**
⇒ FToPolygons
⇒ SelectByIntersection
- **New:**
⇒ ClipSilkscreen

9.8.70:

- **New:**
⇒ GetLayerDimensionsEx

9.8.50:

- **Changed:**
⇒ ImportZeroLengthDraw: new possible value "Ask". The two previous values "Pads" and "Ignore" are still valid.
- **New:**
⇒ PolarityCombineAtLayer
⇒ PolaritySeparateLayer

9.8.20:

- **Bugs fixed:**
⇒ GetSelectionTools: did not correctly interpret the tool type argument.
- **New:**
⇒ SaveCompareLayersErrors

9.8.5:

- **New:**
⇒ AlignToEndpointsOnly
⇒ BuildCustomFromFrame
⇒ RoutingArcsToChords
⇒ RoutingChordsToArcs

9.8.0:

- New:
 - ⇒ GetDCodeElementCounts
 - ⇒ InsertAsPaint
 - ⇒ PolygonsToStrips
 - ⇒ RoutingConnectorArcs

9.6.91:

- New:
 - ⇒ ExePathName

9.6.43:

- New:
 - ⇒ OptimizePaintScratch

9.6:

- Changed:
 - ⇒ .ImportAMRotation

A new item has been added to this property (in addition to “CW” or “CCW”). The new item can take one of three values: “Primitive”, “Aperture” or “Automatic”. These set the option found in “File/Import/Gerber/Options/RS274-X Options/Rotate around”. The “Automatic” setting is recommended. Existing scripts should not be affected by this change.
- New:
 - ⇒ .SelectCoveredElements
 - ⇒ .SelectDuplicateElements

9.4.73:

- Changed:
 - ⇒ ImportZip

9.4.6-65:

- Changed:
 - ⇒ CalculateCopperArea
- New:
 - ⇒ CAMTEKINI
 - ⇒ CAMTEKParameters
 - ⇒ CAMTEKWriteReference
 - ⇒ CompareTwoLayersEx
 - ⇒ DragFrame
 - ⇒ DrillImportOptions
 - ⇒ ImportDiscardPaths
 - ⇒ LastWarningNumber
 - ⇒ Marker

⇒ OnDone

9.4.1:

- **Changed:**
 - ⇒ **Import and Export.**

A new file type has been added: “PentaLogix”. This is a new ASCII format for transfer of geometrical information about elements in the database. The new format is described in Appendix A of the “CAM_ActiveX” manual.
- **New:**
 - ⇒ CreateRaster
 - ⇒ CursorRelative
 - ⇒ PolaritySeparateAfterImport
 - ⇒ SelectClosestAndInfo
 - ⇒ SelectLayers
 - ⇒ ToolNumbersTranscodeAscending

Scripting 9.2:

- **New 9.2.41:**
 - ⇒ GetProcessID
 - ⇒ MergeColinearTraces
 - ⇒ RoutingGroupToPaths
 - ⇒ RoutingUngroupToTraces
 - ⇒ SelectedRoutingPaths
- **New 9.2.27:**
 - ⇒ GetLayerPolarityChanges
 - ⇒ PolarityCombineAll
 - ⇒ PolaritySeparateAll
- **New 9.2:**
 - ⇒ ReplaceSelectedPolygonsWithOverlaps
 - ⇒ SelectUnusedPads
 - ⇒ ConvertSelectionToPolygons
 - ⇒ ReorderLayers

Scripting 9.0.50:

- **New:**
 - ⇒ SelectedArcsToQuadrants
 - ⇒ SetupDCodesDialog
 - ⇒ Changed:
 - ⇒ SetSaveOptions

Scripting 9.0:

- ⇒ A new way to load and run a script file when starting CAMMaster has been added:
CAMMaster /run <name of script file>
where “<name of script file>” is the path of the script that you want loaded and executed. If “/run” is omitted, the script file will be loaded but not executed.

- **New:**
 - ⇒ CopperAreaOfSelection
 - ⇒ CurrentCustomProperty
 - ⇒ CustomProperties
 - ⇒ CustomProperty
 - ⇒ FrameVisible
 - ⇒ OnlyCurrentCustomProperty
 - ⇒ SelectArc
 - ⇒ SelectTrace
- **Changed:**
 - ⇒ GenerateOutlines: now generates polygons rather than F-type trace contours.
 - ⇒ StepAndRepeatUngroup returns TRUE or FALSE depending on the outcome of the operation.

Scripting 8.4.10

- **New:**
 - ⇒ ZoomShowAll_Include0

Scripting 8.4

- **New:**
 - ⇒ AddSnowmanFilleting
 - ⇒ AddTeardropFilleting
 - ⇒ ChordsToArcs
 - ⇒ ClipSilkscreenTraces
 - ⇒ DisplayFilledElements
 - ⇒ DRCErrorsFile
 - ⇒ DRCMaskToMask
 - ⇒ FillSolid
 - ⇒ GerberOutputSplitPolygons
 - ⇒ PointIsInsideSelection
 - ⇒ SelectedText
 - ⇒ UngroupSelectedText

Scripting 8.2.21

- **New:**
 - ⇒ AutoSnapToElement
 - ⇒ CompareTwoLayers
 - ⇒ ConvertDrawnPads
 - ⇒ ConvertStandaloneDrawnPads
 - ⇒ MakeSelectedPolygonsWellFormed
 - ⇒ SelectByIntersection
 - ⇒ SnapTolerance
 - ⇒ StatusLineMessage

Scripting 8.1

- **New:**
 - ⇒ ImportMoveArcCenter

- ⇒ NetCompare
- ⇒ PrintToBMPFile
- ⇒ REFStartLayer

- **Modified:**

- ⇒ GetLayerElementCount: added count of polygons at end of string

Scripting 8.0.X

- **New:**

- ⇒ HighestPossibleLayerNumber. For this version it will return 255
- ⇒ ImportKeepAllFlashes

- **Bugs:**

- ⇒ The Swell Selected method appeared as SwellSelection in the type library. Also it did not work for negative swell values.

Release History

NOTE: Not all released versions are listed below

Version	Release Date
11.24.1	Feb 18, 2024
11.22.1	Jan 26, 2023
11.20.1	Nov 10, 2021
11.18.1	Sep 17, 2020
11.16.53	Mar 28, 2020
11.16.1	Jun 10, 2019
11.14.1	Jun 6, 2018
11.12.1	Dec 14, 2016
11.10.80	Nov 15, 2016
11.10.9	Sep 29, 2015
11.8.24	Aug 12, 2015
11.8.1	Feb 11, 2015
11.6.0	Aug 14, 2013
11.4.45	Jun 24, 2013
11.4.0	Feb 14, 2012
11.2.78	Nov 30, 2011
11.2.74	Sep 19, 2011
11.2.22	Nov 17, 2010
11.2.1	Sep 10, 2010
11.0.80	Aug 1, 2010
11.0.1	Dec 10, 2009
10.6.71	Sep 28, 2009
10.6.0	Apr 28, 2009
10.4.0	Jan 24, 2009
10.2.71	Jan 10, 2009
10.2.0	Oct 3, 2008

10.0.30	May 30, 2008
10.0.1	Mar 17, 2008
9.8.70	Dec 31, 2007
9.8.0	Sep 10, 2007
9.6.91	May 31, 2007
9.6	Jan 2, 2007
9.4.1	Mar 3, 2006
9.2.41	Jun 30, 2005
9.2	Apr 5, 2005
9.0	Aug 5, 2004
8.4.50	Dec 17, 2003
8.4	Oct 27, 2003
8.2.5	Feb 26, 2003
8.1	Oct 23, 2002

Copyright, Disclaimer and Trademark Information

Copyright: © PentaLogix Inc., 1985-2021. All rights reserved.

You may cite or refer to information published in this document, but you may not reproduce or distribute such information in whole or in part without the prior written permission of PentaLogix.

Nothing contained herein shall be construed as conferring by implication or otherwise any license or right under any patent or trademark of PentaLogix or any third party. No other rights under any copyrights of PentaLogix or any other party are granted herein, except as expressly stated above.

Disclaimer:

Reasonable efforts have been made to ensure the accuracy of the information presented. However, PentaLogix assumes no responsibility for the accuracy of the information. Product information is subject to change without notice. PentaLogix may make improvements and/or changes in the products and/or the programs described in these publications at any time without notice.

Trademarks:

CAMMaster, ViewMate, RoutMaster, ProbeMaster, FixMaster and LAV501 are trademarks or registered trademarks of PentaLogix, Inc.

All products and brand names are trademarks or registered trademarks of their respective holders.