

FlowCAD Webinar

Fertigungsdaten bequem aufbereiten mit Cross Probe zum PCB Editor

14. November 2013

Introduction to VisualCAM/GerbTool

- Complete control over PCB designs
- Visual verification
- Analysis
- Optimization
- Scalable Solution

Scalable Solution

- VisualCAM
- GerbTool Designer
- GerbTool Inspector
- GerbTool Communicator
- GerbTool Viewer
- IPC 2581 Viewer (free)
- ISE Database Viewer (free)

General Feature Highlights

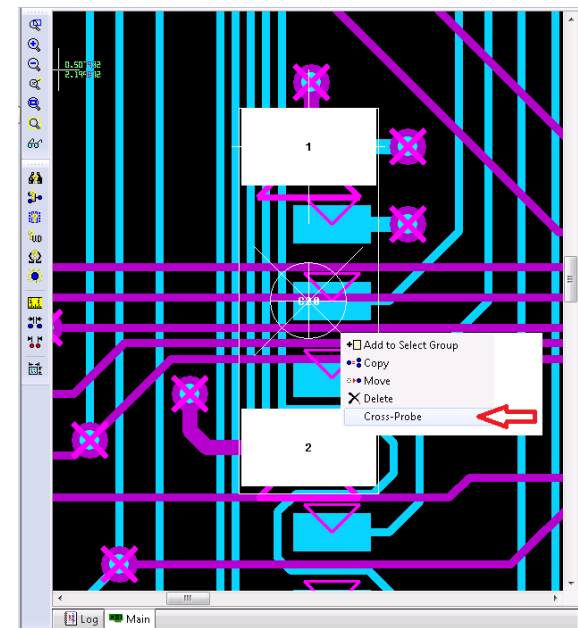
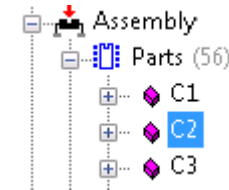
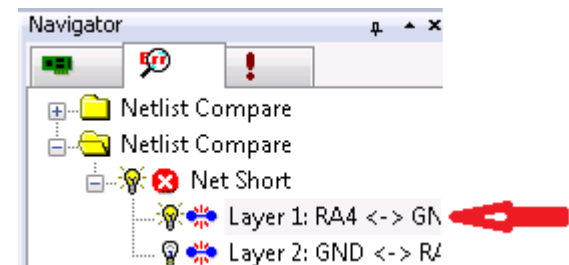
- Intelligent Data Import
- Optimization (Draw-to-Flash conversion, Teardropping, Flatten Composite Layer, Solder/Paste Mask Generator and Optimizer)
- Netlist Comparison
- Dynamic DfM Analysis Suite
- NC Drill & Mill
- Advanced Panelization
- Assembly Reverse Engineering
- Automation and Scripting

What's New in Release 16.4

- **CrossProbe to Allegro/OrCAD (VCAM/Designer)**
- Support for New format IPC-2581 Rev B with many Import/Export enhancements
- Support for New format ODB++ v8
- Support for New Gerber Spec
- Support for Cadence Net_SHORT Property
- Advanced Stencil Design tools with Library support (VCAM and AddOn module to Designer)
- Import / Export Lloyd Doyle AOT format (VisualCAM)
- Multiple Advanced Panelization enhancements
- Clip command enhanced to provide clip by “circular window” with multiple methods
- Clip “outside window” option added
- Pin-Point Error command enhanced

Cross Probe

- Cross Probe using Zoom Commands
- Cross Probe Items using Query Commands
- Cross Probe Analysis Errors
- Cross Probe Items in Main Viewing Window
- Right click an item in the main windows and select Cross Probe or Cross Probe Net.
- Cross Probe from Navigator
- Cross Probe parts and nets from the Navigator

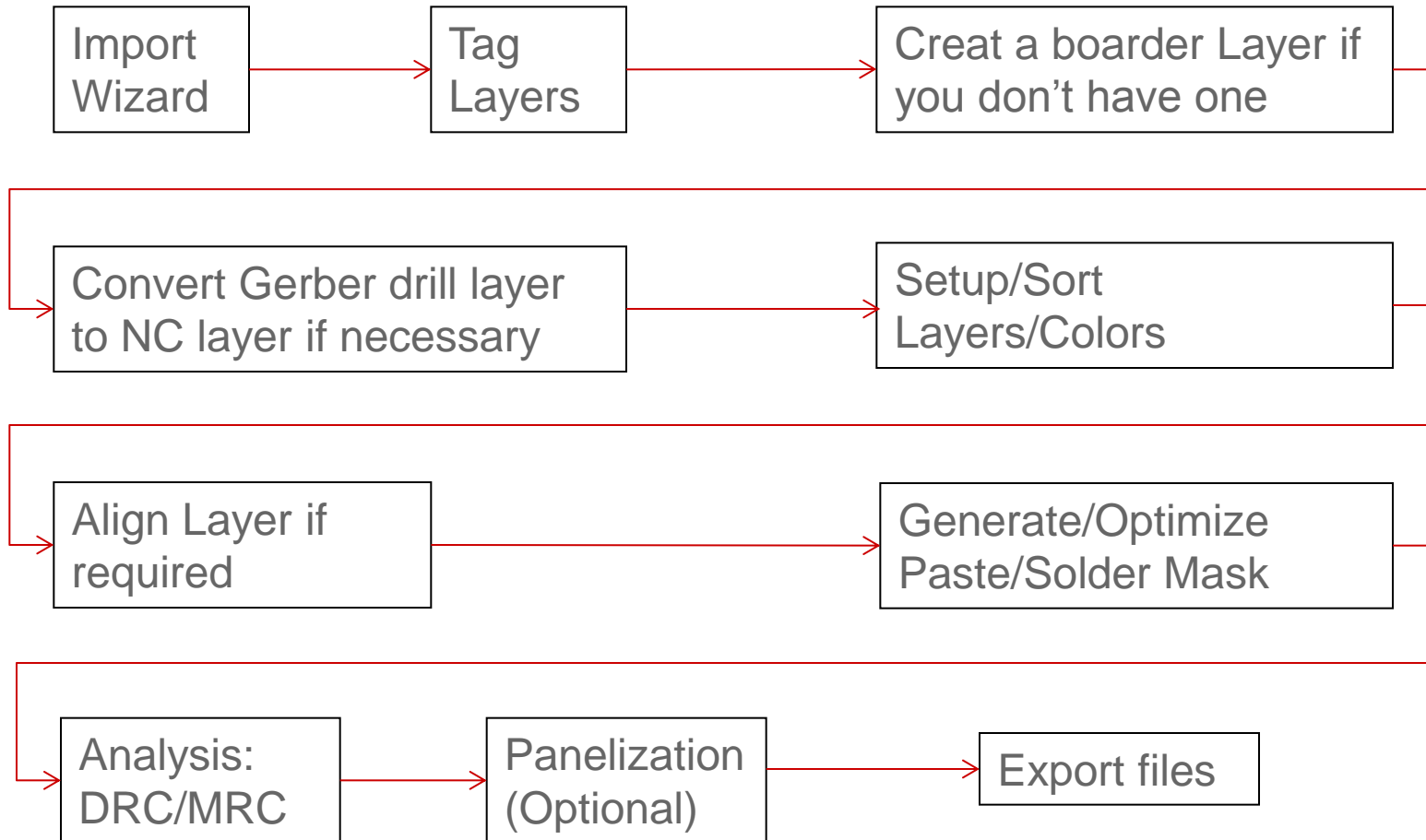


Advanced Stencil Design – Library based

Easily generate IPC compliant paste layers based on automatically identified footprints or define your own footprints and rules for any type of pad configuration

- Identify paste footprints from library information
- Identify paste footprints and add to the Footprint library
- Use IPC-7525B based paste guidelines
- Define custom paste configurations which can be saved from and/or imported into the Footprint library
- Edit the paste configuration for each pad independently within the footprint via paste editor interface
- Generate new paste layers based on the currently loaded Footprint library
- Generate paste analysis reports including: Shape Information, Area Ratio, Paste Volume, Aspect Ratio

Introduction – General Process Flow



Introduction - GUI

Title Bar →

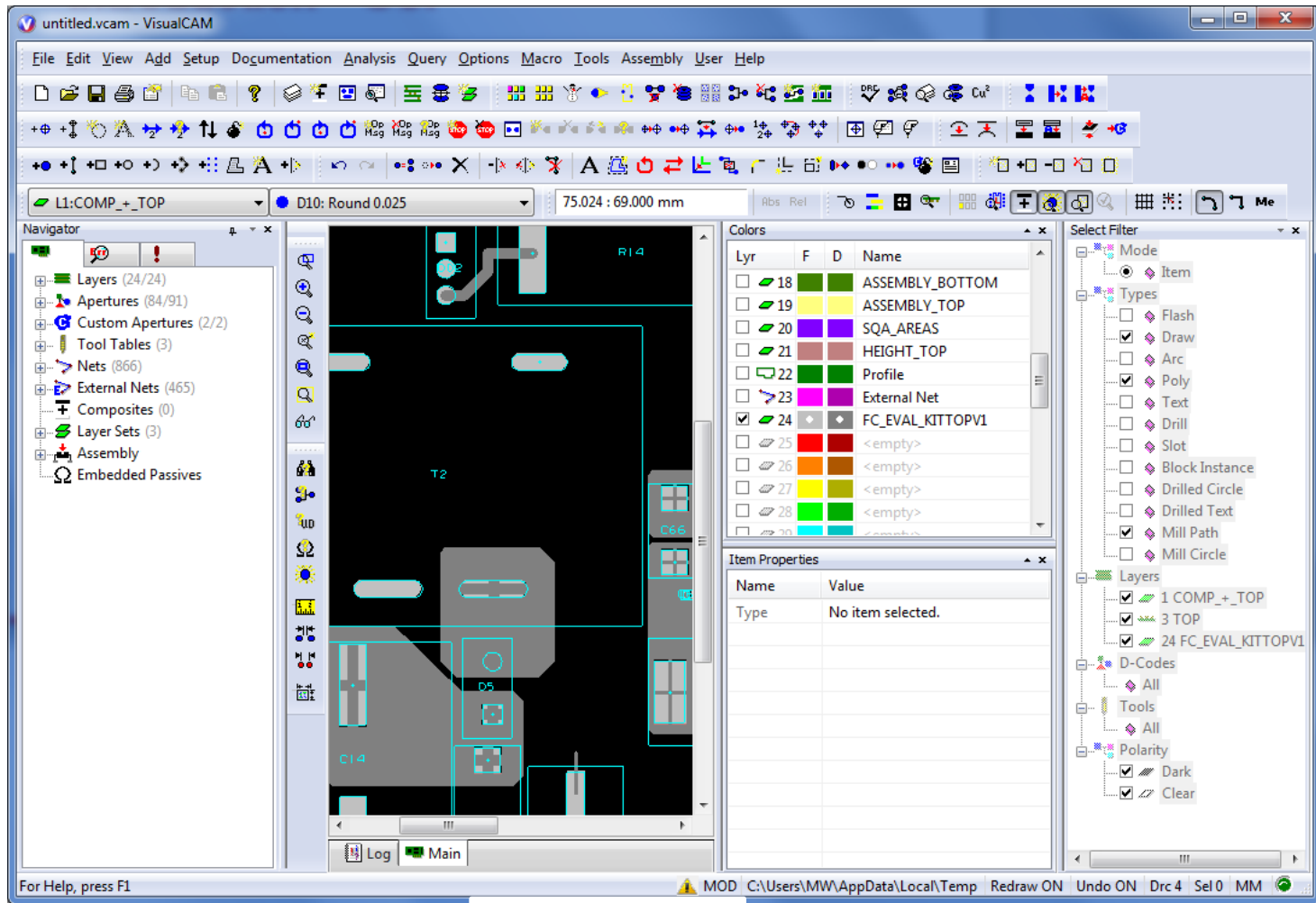
Menu Bar →

Toolbars →

Layer Bar →

Navigator →

Status Bar →



Import Data




Import Wizard

The screenshot displays the 'Import Wizard' in four sequential steps:


































- Step 1:** Introduction screen with instructions to select a folder. The path shown is `E:\Schulungen\GerbTool\Beispiel Vorbereitung`.
- Step 2:** 'Select files to import.' A table lists files with their file types and data formats.
- Step 3:** 'Gerber Import Data Format' dialog for file `FC_EVAL_KITAsse...`. It shows settings for Dialect (RS274X), m.n. (5.5), Terminator, Coordinate Mode (Absolute), Zero Suppression (Leading), Character Set (ASCII), and Special options (Metric, Merge Polygons).
- Step 4:** 'Specify load order of your data files.' A list of files is shown, with `FC_EVAL_KITAsse...` selected.

Filename	File Ty...	Data Format	Length
<input checked="" type="checkbox"/> FC_EVAL_KITAsse...	Gerber	RS274X,5.5,Abs,Metric,Leading	208977
<input checked="" type="checkbox"/> FC_EVAL_KITAsse...	Gerber	RS274X,5.5,Abs,Metric,Leading	317066
<input checked="" type="checkbox"/> FC_EVAL_KITBOT...	Gerber	RS274X,5.5,Abs,Metric,Leading	97634
<input checked="" type="checkbox"/> FC_EVAL_KITGN...	Gerber	RS274X,5.5,Abs,Metric,Leading	95416
<input checked="" type="checkbox"/> FC_EVAL_KITGN...	Gerber	RS274X,5.5,Abs,Metric,Leading	80197
<input checked="" type="checkbox"/> FC_EVAL_KIT11V1.0	Gerber	RS274X,5.5,Abs,Metric,Leading	111875
<input checked="" type="checkbox"/> FC_EVAL_KIT12V1.0	Gerber	RS274X,5.5,Abs,Metric,Leading	94933

Import Data – Layer definitions

-  Top
-  Inner
-  Bottom
-  Plane
-  Silk Bottom
-  Silk Top
-  Mask Top
-  Mask Bottom
-  Paste Top
-  Paste Bottom
-  Border
-  NC (Drill/Mill)
-  Composite
-  Drawing
-  Insulator

Flash Draw

Lyr	F	D	Name
<input checked="" type="checkbox"/>  1			pri.art
<input checked="" type="checkbox"/>  2			int1.art
<input checked="" type="checkbox"/>  3			gnd.art
<input checked="" type="checkbox"/>  4			vcc.art
<input checked="" type="checkbox"/>  5			int2.art
<input checked="" type="checkbox"/>  6			sec.art
<input checked="" type="checkbox"/>  7			slkpri.art
<input checked="" type="checkbox"/>  8			slksec.art
<input checked="" type="checkbox"/>  9			Drill
<input checked="" type="checkbox"/>  10			Outline
<input type="checkbox"/>  11			<empty>

Import IPC2581 Data

Advantage of the IPC2581 Import:

- The layer set up is already correct!
- Drill table is correct
- Aperture's are already defined

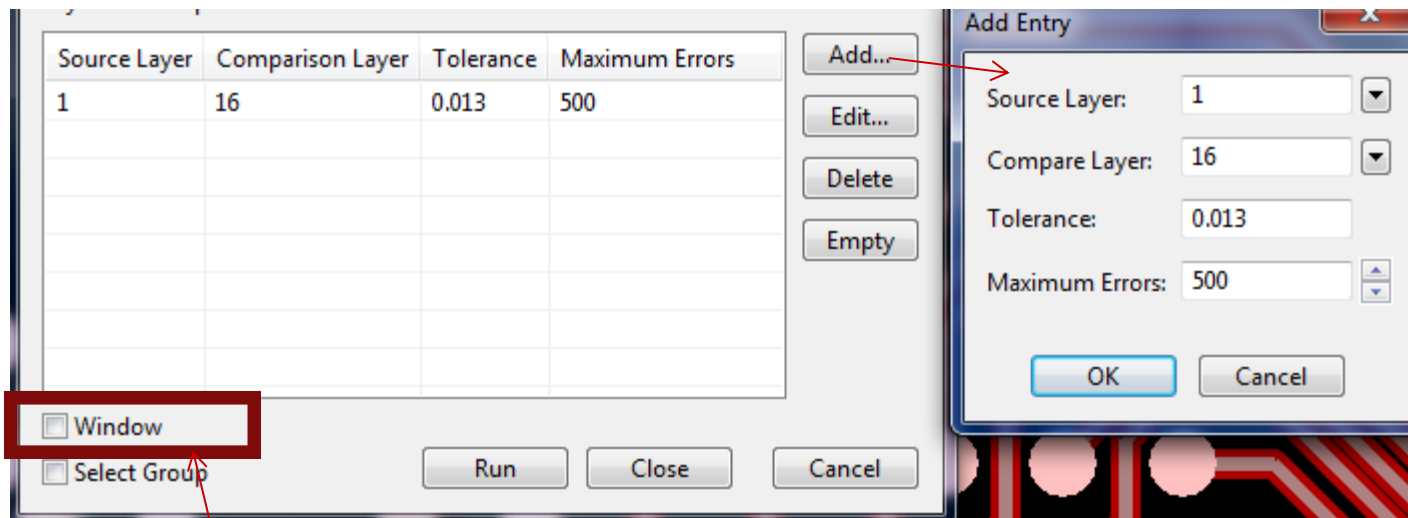
Lyr	F	D	Name
<input checked="" type="checkbox"/>	1		COMP+_TOP
<input checked="" type="checkbox"/>	2		TOP
<input checked="" type="checkbox"/>	3		GND1
<input checked="" type="checkbox"/>	4		I1
<input checked="" type="checkbox"/>	5		POW1
<input checked="" type="checkbox"/>	6		GND2
<input checked="" type="checkbox"/>	7		I2
<input checked="" type="checkbox"/>	8		POW2
<input checked="" type="checkbox"/>	9		BOTTOM
<input checked="" type="checkbox"/>	10		ROUT
<input checked="" type="checkbox"/>	11		COMP+_BOT
<input checked="" type="checkbox"/>	12		DRILL
<input checked="" type="checkbox"/>	13		D_TOP_GND1
<input checked="" type="checkbox"/>	14		D_GND1_I1
<input checked="" type="checkbox"/>	15		D_I1_I2
<input checked="" type="checkbox"/>	16		SQA_AREAS
<input checked="" type="checkbox"/>	17		Profile
<input type="checkbox"/>	18		External Net

DEMO

- Import Data
- Cross Probe

Design Compare

- The Design Compare command compares the Layers of two designs, and reports any differences found



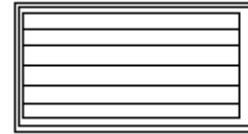
Compare only a selection

Analysis

- The VisualCAM Analysis functions are used to find any flaws in your design prior to manufacturing
 - Using rules that you specify
 - Analyze your design
 - Find errors
 - Determines the best fix for those errors
- Review the errors in the workspace
 - print error reports
 - fix the errors yourself
 - have the system fix the errors for you

Analysis – Preliminary Steps

- Import your data
- Layers have to be “tagged” with the correct Layer type
- Remove any extraneous data on electrical layers
- Convert drawn pads to flashes
- Align Layers if they are misaligned
- Remove redundant pads
- Valid Netlist
 - Import or create one
- Select the Analysis commands and run analysis
- After performing the analysis
 - VisualCAM automatically displays the errors
 - Use Navigator to view and automatically fix the errors



Analysis Type

- Information
- DRC (Design Rule Check)
- DFF (Design For Fabrication)
- NLC (Net List Compare)
- Embedded Passive

Analysis - Information

- Min Air Gap
 - Reports the shortest distance between any two items in the design
- Conductive Layer Count
 - Reports the total number of conductive layers
- Board Size
 - Reports the extents of the Border layer, if one exists
 - Otherwise the extents of the conductive layers is reported
 - drawing and other graphical layers are ignored
- ...

Analysis to get general Information

Analysis - DRC

- Pad/Pad
 - The minimum spacing allowed between pads.
- Pad/Trace
 - The minimum spacing allowed between pads and traces.
- Trace/Trace
 - The minimum spacing allowed between traces.
- ...

Analysis - DFF

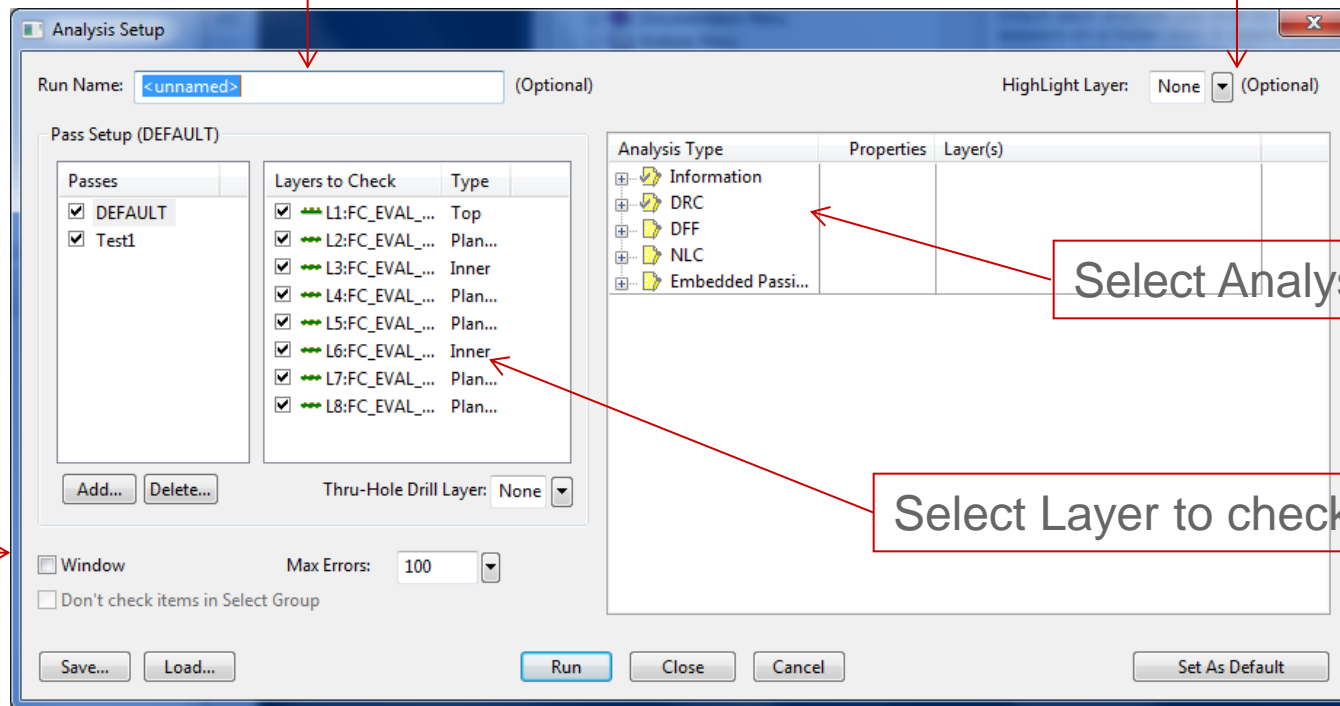
- Acid Traps
 - Area where etching solution accumulates during manufacturing
 - over-etching
- Copper Slivers
 - This are copper areas which are so narrow that they will likely flake off
- Resist Slivers
 - Same as Copper Sliver but on resist area
- Isolated Thermals
 - Over-etching can isolate thermals (on negative plane)
- Starved Thermals
 - Check the number of thermal ties (on negative plane)
- Top / Bot Solder Bridges
 - When a mask layer is too much oversized, the danger of solder bridges increase
 - This test check the minimum mask area within the tolerance
- Pin Holes
 - Too little pin holes can be a cause of acid traps
 - This Test detect to small pin holes

Analysis Settings

Enter a Name

(easier to find the results in Navigator Window)

Copy Error Items to an empty layer



Select Analysis Type

Select Layer to check

Run Analysis on selected Window

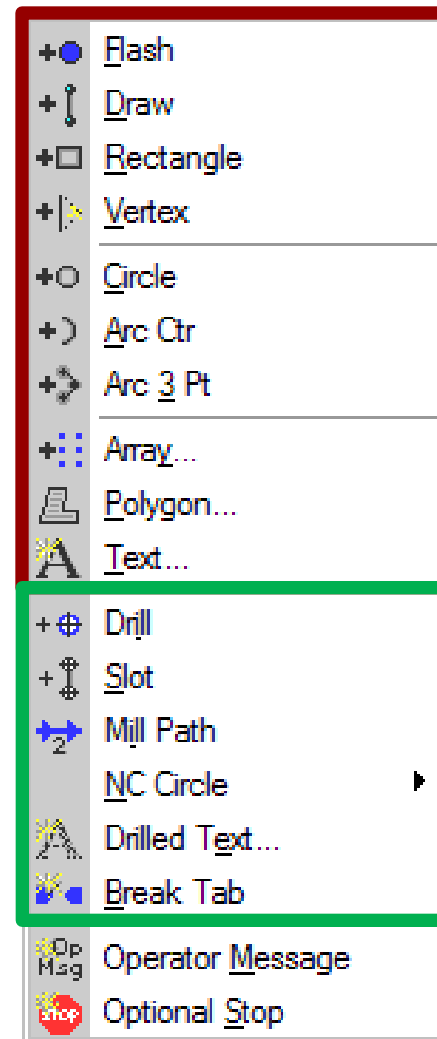
DEMO

- Analysis
- Cross Probe

Modifications on existing Design Artwork

- Add Menu

Copper Utilities



Drill file utilities

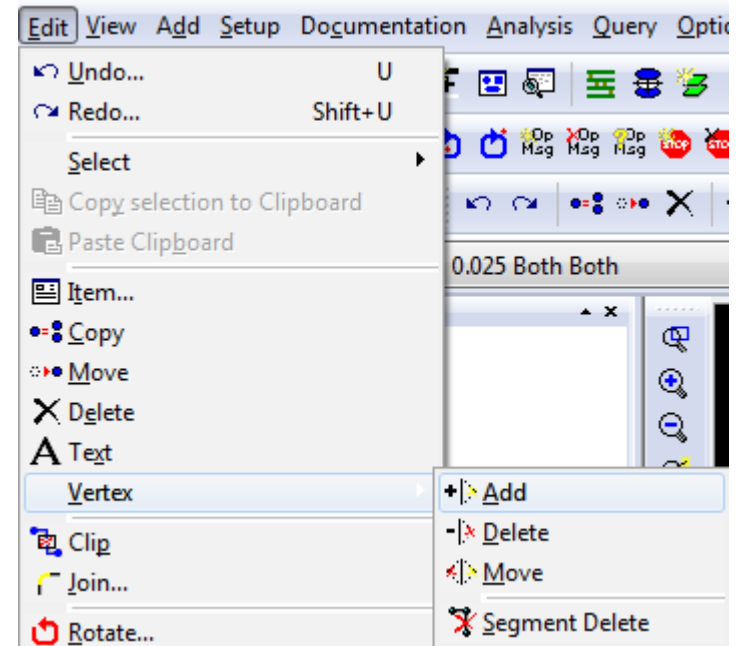
Modifications on existing Design Artwork

- Vertex: Modify existing traces

Before



After

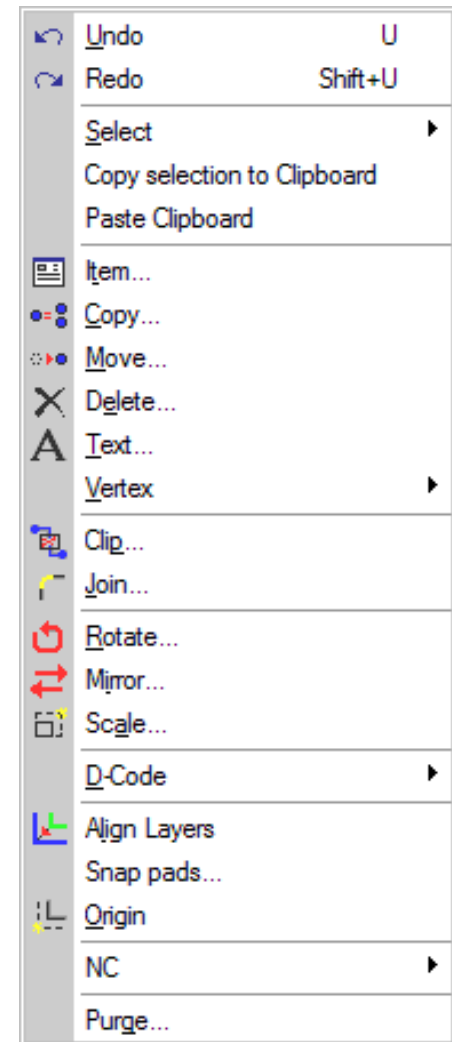


Modifications on existing Design Artwork

- Edit Menu

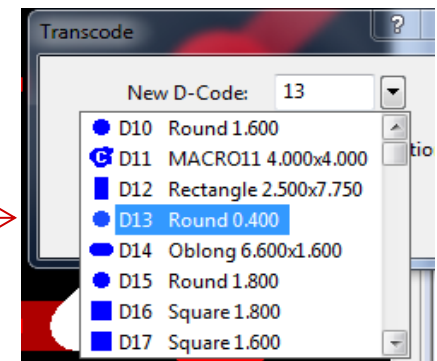
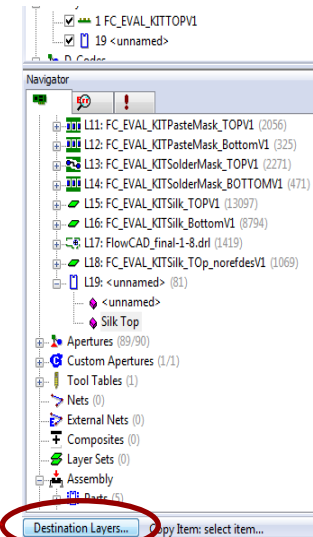
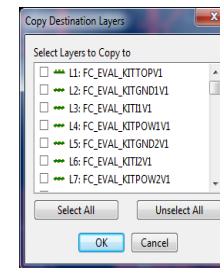
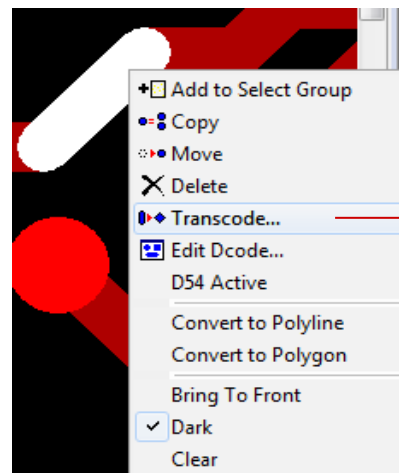
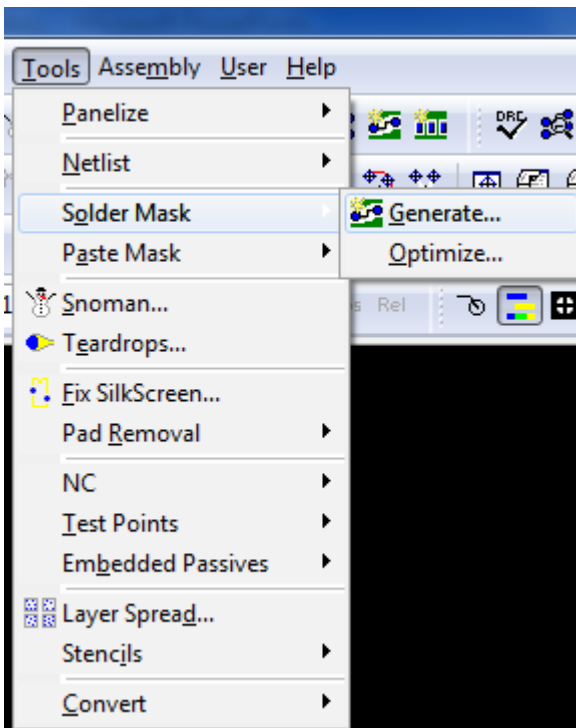
Clip: erase all items in selected area

Join: Connect two segments



Modifications on existing Design Artwork

- Copy to another layer
- Change Trace Width
- Solder Mask / Paste Mask Generation



Panelization

- Panelization is the process of creating an array of your PCB for a manufacturing panel
- Allows multiple PCBs to be created at one time
- Optimize material usage
- Shortening manufacturing turnaround time

Panelization

Simple

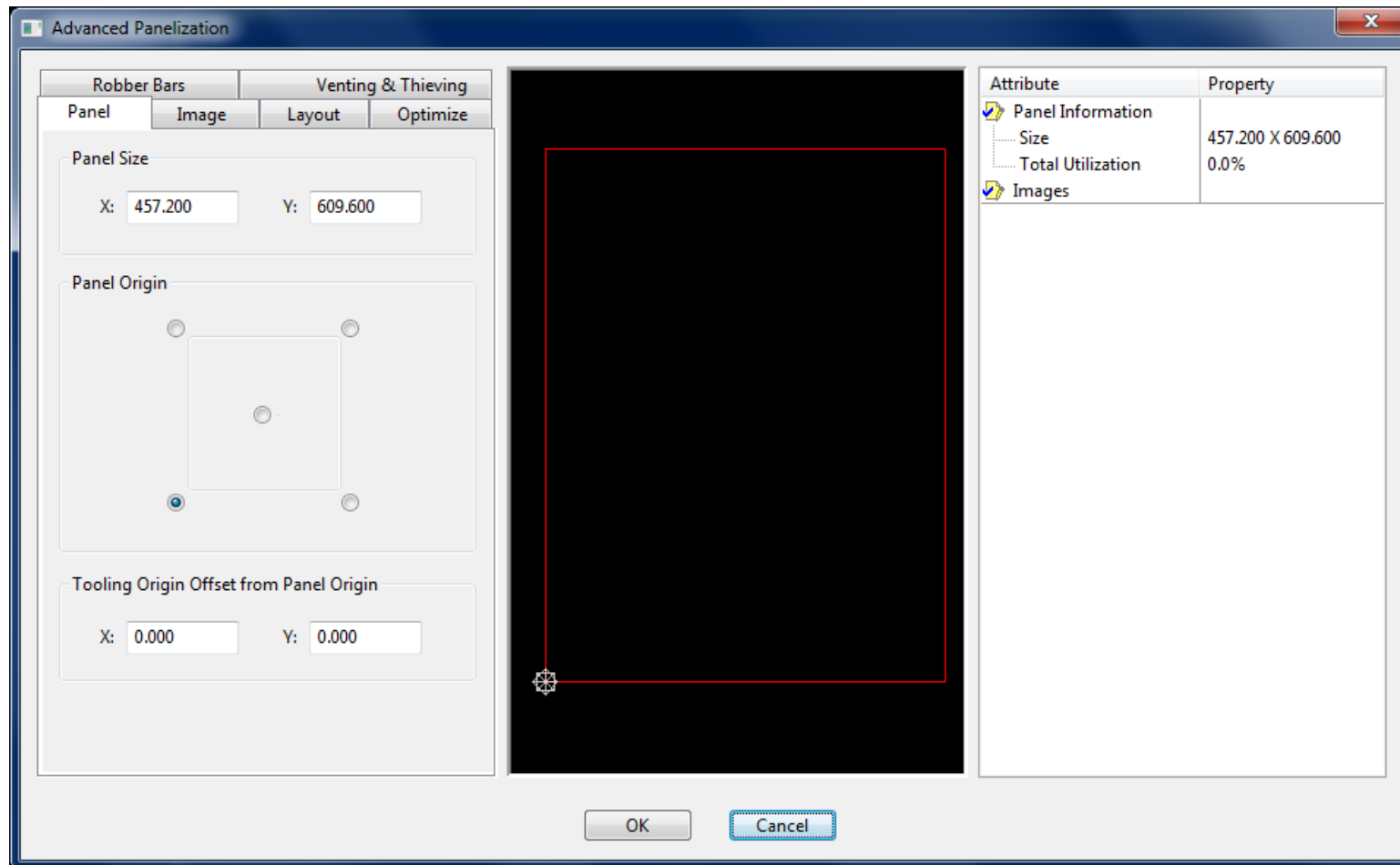
The screenshot shows the 'Panelize' dialog box with the following settings:

- Panelize Method:** Standard (Copies data), Virtual (Use SR codes). Current Panel: [Dropdown], Setup Virtual Panels: [Dropdown].
- Panel Parameters:**
 - Auto Calculate number of copies and placement in Film Box
 - Minimum Spacing Between Images:** X: 20.000, Y: 20.000
 - Copies:** Cols (X): 2, Rows (Y): 2
 - Minimum Spacing to Film Box:** X: 30.000, Y: 30.000
 - Image Offsets:** X: 300.000, Y: 300.000
- Venting:**
 - Auto Vent, Vent to Image Spacing: 0.000
 - Pattern Spacing:** X: 0.000, Y: 0.000
 - Destination:** Layer: 0 [Dropdown], D-Code: 0 [Dropdown]

Buttons at the bottom: Preview, OK, Cancel.

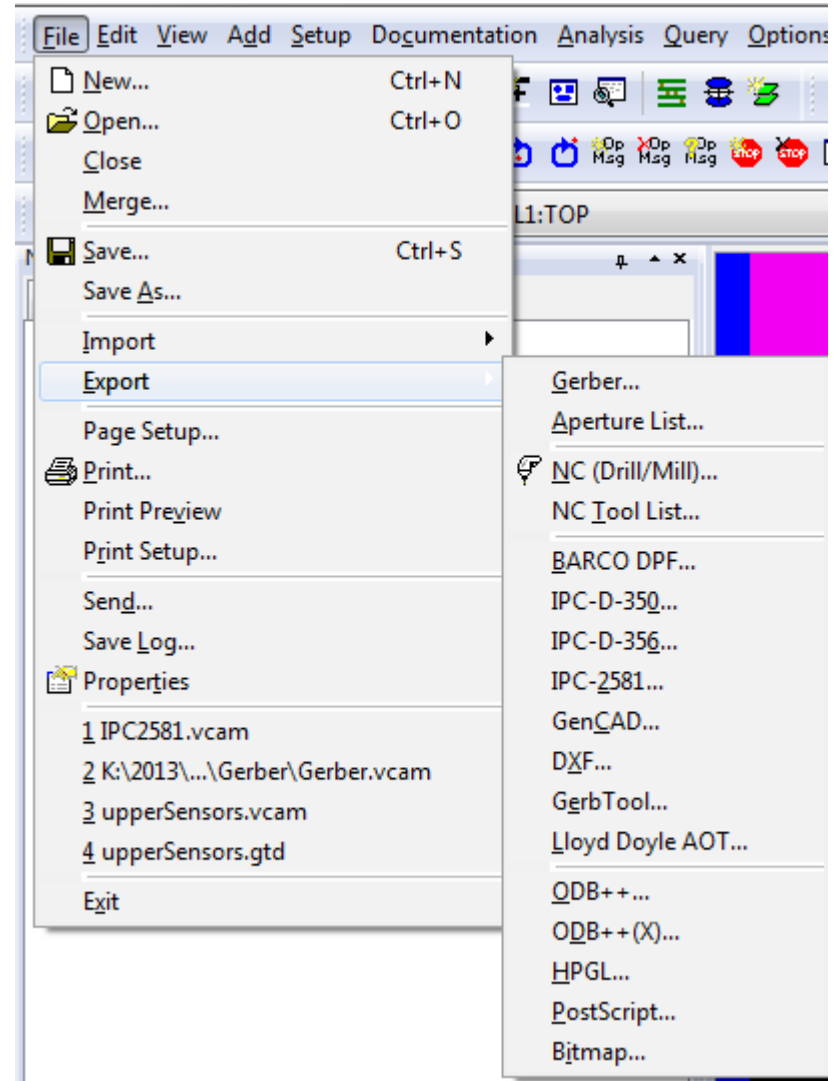
Panelization

Advanced



Export

Different formats are supported



DEMO

- Panelization

End

Thank you for your attention!

**For any questions or feedback, please
contact us:**

Tel.: + 41 (0) 56 485 91 91

E-Mail: support@flowcad.ch

FlowCAD