FloWare Modules – Enhancing Productivity

Version v58
FloWare Features

• General purpose utilities
  – Functions not in standard tool yet
  – Not written for one specific customer
  – Everybody should benefit

• Easy installation
  – Also casual users must be able to install FloWare
  – No variables
  – Menus will be created automatically
  – Integrates perfectly into existing customizations

• Documentation
  – Full documentation is provided for every module
Table of Contents

• Categories
  – **PCB Layout**
    • OrCAD PCB Designer
    • Allegro PCB Editor
  – **Schematic Entry**
    • OrCAD Capture
    • Allegro Design Entry CIS

• **Contact Information**
FloWare Modules
PCB Layout

OrCAD PCB Designer / Allegro PCB Editor
FloWare Modules PCB Editor

- Advanced Mirror
- Advanced Testpoint Check
- AOI Check
- Assign Net to Via
- Barcode Generator
- Batchplot
- CAF-DRC
- Change Width
- Class Color
- Coil Designer
- Cross Copy
- Cross Section Generator
- Custom Variables
- Design Compare
- Drafting Utilities
- Drawing Designer
- Drawing Size
- FPGA Utilities
- Highlight Dummy Pins
- IBIS Prototype Modeler
- Label Generator
- Label Tune
- Mask Generator
- Net Color View
- Padstack Finder
- Padstack Usage
- Panelization
- PCB Library Plot
- Polar Grid Utilities
- Post Processing
- Push to Grid
- Quick Symbol Edit
- Replace Via
- Shape Utilities
- Shield Generator
- Shield Routing
- Silkscreen
- Snap Generator
- SVG Export
- Synchronize Testprep
- Variant Assembly
- Variant BOM
- Z-DRC

Videos available on YouTube
Advanced Mirror

- Enables mirror operations while moving or copying elements
  - Mirror across subclasses (Geometry and Layer)
  - Mirror on the same subclass (Geometry only)

- Including placement and routing

- Selection through window or single pick, dynamic preview

- Special handling for symbols, vias and text in Geometry only mode
Advanced Testpoint Check

- Addresses various rules for testpoint checking
  - Testpoint to Testpoint check
  - Testpoint to Component check taking component height into account
  - Visualization of restriction areas
  - DRC marker generation
  - Constraints reuse through configuration files
Advanced Testpoint Check

- Spacing rules may be specified
  - As fixed spacings
  - Using rule sets, which account for component height
AOI Check

• Helps users to check AOI related rules directly in **PCB Editor**

• Shadowing can cause serious issues in verification process
AOI Check

- Shadows can be calculated in various directions based on specified camera angles taking component height into account.

- Special rules apply to 3D inspection systems.
Assign Net to Via

- Enables users to change the net on existing vias
- No need to delete and route new ones
- Use model
  - Select net from list (including wildcard support) or by context **RMB > Pick Net**
  - Click on one or more vias
  - Choose **RMB > Done** to confirm
Assign Net to Via

Example

Shape GND

Vias to be connected
Barcode Generator

- Prints barcodes on a PCB layer as part of the manufacturing process
- Supporting definitions Code 39, Code 128, QR Code and Data Matrix
- Adjustable parameters corresponding to selected code
- Additional drawing options (e.g. show text, inverted display)
- Dynamic preview during parameter change
- Parameter preset through configuration file
Barcode Generator
Batchplot

- Creates documentation
  - Fully automated
  - Multipage or single page files
  - Visibility control based on views (color views as well as artwork views), similar to visibility panel
  - Order of view items can be changed by user
  - Plotsets allow grouping views into separate PDF files
  - Load and save configuration to disk
- Three modes
  - PDF Export from Allegro / OrCAD PCB Editor
  - HTML Export
  - Batchplot (legacy)
Batchplot

- Multiples viewsets can be defined
- A viewset may contain
  - Film records
  - Color views (!!)
- Each viewset corresponds to one PDF file to be generated
  - Name of the PDF file can be specified (output)
- Each viewset may reference a profile to be used during PDF export
  - E.g. black white print only
  - With / without meta data
  - …
- Single push button to generate all data …
**Batchplot**

Setup Viewsets for PDF Export

- Use context menu in tree view
  - Rename
  - Edit
  - Add
  - Delete
- Edit views
  - Select film views
  - Select color views
  - Change order
**Batchplot**

**Other Modes**

- **HTML Export**
  - Generates an HTML report including SVG graphics
  - Setup regarding viewsets and profiles similar to PDF Export

- **Batchplot (legacy)**
  - Selecting Setup will launch the existing / legacy application
  - Still available and supported
Batchplot

Final document opened with Acrobat Reader
CAF-DRC

- Special DRC for Conductive Anodic Filaments (CAF) on PCBs
- Performs Hole-to-Hole clearance checks along the fiber structure
- Based on voltage classification
- Adjustable fiber orientation
CAF-DRC

• Use model
  – Classify nets by voltage property or by spacing net class
  – Launch application
  – Specify rules and select mode to be used
  – Export / import rules for reuse purposes
  – Run DRC
CAF-DRC

• Results
  – DRC markers in PCB Editor and Constraint Manager including crossprobing
  – Visualization of DRC polygons on DRAWING FORMAT subclasses CAFDRC_DV1, _DV2, DV3 and _DV4
  – Handles all drill and slot types
Change Width

- Change the width of clines and clines segments
- Supporting a filter mechanism in that the changes are only applied to segments matching a given width
- Select by Pick, Window, Temp Group or Find By Name
- Highlight and report functionality including cross probing
Class Color

- Enables users to color spacing net classes for review and documentation purposes
  - Color chooser
  - Legend support
  - Settings stored in database
  - Export / import settings
Coil Designer

- Includes useful functions when designing planar spiral inductors on a PCB
- Support for different spiral pattern
  - Round
  - Rectangle
  - Octagon
  - Hexagon
- Features
  - Parameterized input in terms of width, spacing, number of turns, etc.
  - Support for rotation, flipping and scaling
  - Dynamic preview during parameter change
Coil Designer

Basic spiral patterns

**Round**

**Octagon**

**Rectangle**

**Hexagon**
Coil Designer

Rectangle spiral with corner modifications

Mitered corners

Rounded corners
Coil Designer

- Other features
  - Realization as a combination of cline segments or one single shape
  - Addition of padstacks to the start / end points
  - Automatic generation of route keepouts in the inner area
Cross Copy

• Problem
  – Customer needs to move objects from one class / subclass to another
  – Command Edit > Copy does not support a destination layer
  – Customer gets error messages, e.g. E – Changing shape to a different class is not supported
  – Customer has to find workarounds (e.g. export subdrawing, text edit clipboard file, import subdrawing)

• Solution
  – Cross Copy closes the gap
  – Can move or copy objects to any destination layer for various purposes
  – In contrast to Create Detail, objects are not destroyed (e.g. text will stay text!)
Cross Copy

- Advanced features
  - Shape processing
    - Include / exclude voids
    - Fill or decompose with / out voids
    - Unfill shape
  - Clines / lines processing
    - Can be converted to shape
    - Various endcap types
  - Padstack processing (pins and vias)
    - Drawn as shape on destination layer
    - All visible layers are merged
    - Resulting pad shape can be voided by drill hole information
  - Merging support
    - Selected data (find filter settings) is merged
Cross Copy

Example: Merging data to soldermask shape

Selection

Result

Vias

Drills

Clines

Pins

Voids

One (!) merged shape
Cross Copy

Example: Converting pseudo shapes (stroked lines) to real shapes

Note: Typical situation after importing third party gerber data into Allegro – copper needs to be reanimated!!!
Cross Copy

Example: Backup data on Board Geometry / Assembly_Notes for documentation purposes
Cross Section Generator

- Enhanced support for rigid-flex applications
- Customizable content
  - Custom charts with more meaningful visualization
  - Layer attributes, order, ...
  - Colors, fill styles, ...
  - Vias, stacked vias, backdrill and embedded status support
- SVG export
  - Export graphics for documentation purposes
- Configuration stored in db
- Automatic update
Cross Section Generator

- Content can be defined for each stackup from cross section individually
- Also symbols can be placed individually
Cross Section Generator

- A custom chart lets you combine stackups in an arbitrary order.
- Feature many customers were asking for ...
- Additional information (vias, stacked vias, etc., can be applied to each stackup item individually)
Cross Section Generator

• Backdrill support
Cross Section Generator

- Embedded component support
Cross Section Generator

- Various drawing options
Cross Section Generator

- Export SVG
  - Mainly for documentation purposes
  - Separate color profile can be defined
Custom Variables

- User defined variables in PCB Editor, useful for title blocks, etch texts etc.
- Similar to Design Entry HDL
- Automatic update of variables across all subclasses
- Variant support
- History capabilities
- Variable values can be sourced from external control file or even cpm-file
Custom Variables

Define Placeholders

Update Example

Parameters
Variable name: PART_NUMBER
Display value: \{PART_NUMBER\}
Destination layer: Drawing Format, Title_Block

Attributes
Block: 3, Rot: 0.0000, Align: Left, Mirror: No

Advanced
Type: Standard
History: None

Mode
- Add placeholder
- Modify placeholder
- Delete placeholder
Design Compare

• Compares two databases and identifies the differences

• Useful when tracking changes in the product lifecycle

• Two modes
  – **Standard Compare**
    Generates an HTML report with differences that apply to netlist, placement, BOM, testpoints, etc.
  – **Graphical Compare**
    Based on IPC2581 all or individual layers can be compared graphically, differences will be highlighted, markers can be generated
Design Compare

Standard Compare

- HTML based report
- Expand / collapse functionality facilitates navigation
# Design Compare

## Graphical Compare

![Graphical Compare Example](image.png)

## Design Compare Main Panel

<table>
<thead>
<tr>
<th>Layer</th>
<th>Type</th>
<th>Differences</th>
</tr>
</thead>
<tbody>
<tr>
<td>TOP</td>
<td>Etch</td>
<td>Not processed</td>
</tr>
<tr>
<td>GND</td>
<td>Etch</td>
<td>Not processed</td>
</tr>
<tr>
<td>VCC</td>
<td>Etch</td>
<td>Not processed</td>
</tr>
<tr>
<td>BOTTOM</td>
<td>Etch</td>
<td>Not processed</td>
</tr>
<tr>
<td>IPC2581DRILL_1-4</td>
<td>Etch</td>
<td>Not processed</td>
</tr>
<tr>
<td>SOLDERMASK_TOP</td>
<td>Mask</td>
<td>Not processed</td>
</tr>
<tr>
<td>SOLDERMASK_BOTTOM</td>
<td>Mask</td>
<td>Not processed</td>
</tr>
<tr>
<td>PASTEMASK_TOP</td>
<td>Mask</td>
<td>Not processed</td>
</tr>
<tr>
<td>OUTLINE</td>
<td>Misc</td>
<td>0</td>
</tr>
</tbody>
</table>

**Design Info**
- V1: amplifier_v1.brd
- V2: amplifier_v2.brd

**Display Control (Selected only)**
- V1: On/Off
- V2: On/Off
- Diff: On/Off

**Miscellanea**
- Limit Check to outline extents only

**Min Aperture**: 0.005
**Min Area**: 0.025
Drafting Utilities

- Includes useful functions for drafting purposes
- Functions support dynamic preview before changes are committed
- Several functions
  - Modify line
  - Modify arc / circle
  - Copy array line
  - Copy array arc / circle
  - Trim objects
  - Cut objects
  - Arc2Vector
  - Ellipse (New)
Drafting Utilities

Modify line

- Change line attributes on a parameter basis:
  Coordinates as well as angle, width and length
- Angle display can be swapped
- Fix point for rotation and length changes can be chosen
- Fine-tuning capabilities for angle, width and length

![Diagram showing line attributes and angle display options.](image)
Drafting Utilities

Modify arc / circle

– Change arc or circle attributes on a parameter basis:
  Center coordinate as well as radius, start angle,
  sector angle and width
– Fine-tuning capabilities

![Modify arc / circle diagram]
Drafting Utilities

Copy array line

- Copies lines in parallel to a selected object
- Number of copies can be specified as well as air gap (or center gap respectively)
- Swap side
Drafting Utilities

Copy array arc / circle
- Copies arcs in a concentric way to a selected object
- Number of copies can be specified as well as air gap (or center gap respectively)
- Direction can be chosen
Drafting Utilities

Trim objects

- Trims object endpoints to calculated intersections
- Works with arcs and line objects
- Single point mode when two intersections are found (in case of arcs), ability to swap
- Trim vertex for both objects, first only or second only

![Diagram showing the effect of trimming objects]
Drafting Utilities

Trim objects: Single point mode

Original objects

Result
Drafting Utilities

Trim objects: Trim vertex from …
Drafting Utilities

Cut objects
- Enables users to cut line, arc or circle objects for various purposes
- Two commands
  • Cut objects > By selection
  • Cut objects > Draw path
- Useful when layout portions need to be isolated before moving them to a new location
Drafting Utilities

Cut objects – By selection

Cut objects – Draw path

Minimum cut width limited by database resolution
Drafting Utilities

Ellipse
- Useful for pad shapes and RF applications
- Axis parameter $a$ and $b$
- Geometry
  - Full
  - Partial (specify start, end angle)
- Type
  - Solid filled shape
  - Draw line path
- Number of segments
Drawing Designer

- Useful when creating manufacturing drawings
- Records the creation of scaled views (including mirroring and rotation)
- Automatic updates
- Configuration stored in database
- Configuration export and import
Drawing Designer

Top side

Bottom side

Scale 2:1

Scaled views

Board
Drawing Designer

- Ministatus panel offers various options when creating new drawing items
  - Destination layer where data is written to
  - Scale, rotation, mirror of the item
  - Pad style: filled, unfilled
  - Shape: As defined, unfill all, hatched

Example:

Note: Hatch option only applies to solid filled shapes
Drawing Size

- Let’s you change the design extents in a quick and easy way
- Useful if you want to reduce design extents to minimum values
- Settings can be applied to all or individual sides
FPGA Utilities

- Check and report any pin swapping which has been done in a PCB layout
- Export pin constraints to csv and vendor specific formats
  - Altera, Xilinx, Actel, Lattice
- Works in both flows
  - Allegro Design Entry HDL
  - OrCAD Capture
- Various options
FPGA Utilities

Differences Report

- Pin Constraints for Xilinx XDC
- Design: D:/demo/placed.brd
- Date: Apr 05 15:41:47 2018

- Pin Constraints for Altera Quartus
- Design: D:/demo/placed.brd
- Date: Apr 05 15:43:23 2018
Highlight Dummy Pins

- Assigns a color to all pins in a design which are not connected to any net
- Including report and crossprobe functionality
- Useful for review purposes
IBIS Prototype Modeler

- Creates scalable IBIS models for early stage signal integrity analysis

- Features include
  - Ability to create Push / Pull, Open-Sink and Open-Source buffers
  - Adjustable voltage levels
  - Adjustable internal resistance including a calculator
  - Adjustable slew rate \( dV / dt \)
  - Ability to include saturation and clamping effects
  - Support for single-ended as well as differential outputs
  - Support for output only and bidirectional models
  - Ability to view curves in Sigwave
  - Generate DML models as well as native IBIS models
IBIS Prototype Modeler

- Output only
- Bidirectional
- Single ended
- Differential

Pullup, Pulldown and Rload curves

Power, Ground clamp
Label Generator

- Creates signal name and pin labels for documentation purposes
- Selectable label content (e.g. signal same + pin number)
- Individual label origin (e.g. pin, or via origin, but also cursor intersecting cline)
- Aligned text rotation mode (text angle value follow cursor gesture)
- Supports three different object types (pin, via, cline)
- User defined text size, justification and rotation
Label Generator

Procedure
Label Generator

Label Content

**Display Content:**

- **NET_NAME**
- SYMBOL_PIN_NUMBER
- PIN_NUMBER_NET_NAME
- NET_NAME_PIN_NUMBER
- NET_NAME_PIN_NUMBER
- PIN_NUMBER
- NET_NAME

**NET_NAME**

- RD3
- RD2
- RD1

**PIN_NUMBER_NET_NAME**

- [U6.84]RD3
- [U6.83]RD2
- [U6.82]RD1

**PIN_NUMBER**

- U6.84
- U6.83
- U6.82
Label Tune

• Adjusting component labels automatically
  – Better readability
  – Saves a lot of time when creating assembly drawings

• Features
  – Label can be chosen (e.g. Refdes, Value, Part Number, …)
  – Automatic rotate, center, fit
  – Automatic mirroring
  – Visibility control
  – Various options …
    • Max block size, max height
    • Boundary clearance
Label Tune

Typical situation in PCB Editor

Result after running Label Tune

Animation
Label Tune

Another example
Mask Generator

• In some cases customers may want to oversize mask data (soldermask, pastemask) due to manufacturing requirements

• Data will be generated on manufacturing subclass. The original padstacks won’t be touched

• Input parameters
  – Source data ("What to oversize") can be specified separately for vias and pins
  – Oversize values (positive and negative)
  – Side to process
  – Manufacturing subclass where data will be written to
Mask Generator

Pins and vias on TOP

Manufacturing subclass MSK_OV30U_TOP
Net Color View

- Allows saving and restoring net color and rat visibility settings
- Useful for floorplanning and route feasibility studies
- Global visibility commands
  - Useful hands-on for configuring a view (display / blank rats and assign net colors)
  - Find By Name support
  - No need for jumping and travelling to standard commands from PCB Editor
Padstack Finder

- Enables users to search and navigate for padstacks in the current design
  - Filter capabilities
  - Distinguish between pins and vias
  - Padstack name
  - Drill size
  - Wildcard support
  - Highlight and crossprobe
  - Color by group
Padstack Finder

• Now supporting Vias Stacks
  – Display label formed by individual drill spans
  – Highlight and crossprobe
Padstack Usage

• Generate padstack reports by extracting data from PCB Editor symbol libraries (*.dra). Different reports are available:

• Report types
  – Where-Used
    Lists all footprints that use a given padstack
  – Padstack Standard
    Lists the padstack definitions for a given footprint
  – Padstack Detailed
    Lists detailed information for a given footprint. This information includes number of pins, vias and mechanical pins, pin numbers, xy coordinates, etc.

• Report format
  – HTML
  – Excel
  – Text
Padstack Usage

Report examples
Panelization

- Simplifies panel documentation process
- Boards can be stepped individually or by array
- Boards can be rotated and / or mirrored individually
- Automatic update
- Automatic notification if boards have been modified
- Based on proven mdd-Technology (Design Reuse, Place Replicate)
- Supports standard panels as well as family panels
- Configuration stored in database
Panelization

- **Setting up a new panel**
  - Open a new database or panel template
  - Launch **Floware > Panelization**
  - Link your boards
  - Create modules
  - Place a single instance or an array of your board
  - Done

- **Updating an existing panel** (e.g. original layouts have been modified)
  - Open the panel database
  - Launch **Floware > Panelization**
  - Choose Update panel
  - Done
  - Create manufacturing data as usual …
Panelization

- Interactive placement
  - Place single instances or array
  - Dynamic preview attached to cursor (board size, orientation, rotation, mirror)
  - Display design bounding box or actual outline
  - Snap to grid option

Display bounding box

Display actual outline
Panelization

• Panel modifications
  – Change parameters (e.g. x, y) of instance to be changed
  – Fields become yellow
  – Choose context menu Update changes to confirm
Panelization

- Family panels
  - Contain more than one board layout
  - Simply link the board databases
  - Make your placement
Panelization

- Panel status
  - Automatic notification when board databases have been modified
  - Status report gives details about time stamps
PCB Library Plot

- Documentation of PCB footprints within a selected library
  - Includes graphical representation of footprint graphics
  - Graphics will be scaled into dedicated areas for better readability
  - Additional attributes (size, pitch, padstacks used etc.) are automatically extracted and written to the document
  - Frame templates are provided
  - Customizable contents and drawing styles including drill holes
  - One single PDF document for all footprints
PCB Library Plot

Default Mode

• Basic information
• More footprints per page

Detailed Mode

• Additional information
• Less footprints per page
PCB Library Plot

- Templates
  - Define the areas to be used for plotting which correlates to the number of footprints per page
  - Metadata support for title blocks (page number, date, user, library name)
  - Default templates provided
  - Customer can define his own templates
PCB Library Plot

Examples

6 footprints on A3
Detailed attributes

4 footprints on A4
Basic attributes
PCB Library Plot

Customizable content including drill holes and various drawing styles

Assembly TOP including etch and drill holes

Placebound as hatch shape including etch and drill holes
Polar Grid Utilities

• Useful for boards where circular placement and routing is required

• Providing a set of functions
  – Defining and editing a polar grid
  – Polar placement
  – Polar routing
  – Polar shape editing including voids

• Snap control
  – Coarse and fine grid
  – System grid

• Polar grid can be also used in conjunction with standard PCB Editor commands using RMB > Snap pick to > Intersection
Polar Grid Utilities

- **Polar Grid Setup**
  - Define a new grid
  - Edit an existing grid
  - Fully parameterized
  - Coarse and fine grid settings
  - Line and dot style support
  - Stored as format symbol in database
  - Visibility control
Polar Grid Utilities

• Polar placement
  – Special mode for placing component in circular or radial fashion
  – Snap to fine or coarse grid
  – Various alignment and rotation capabilities
  – Spinning on polar grid basis (e.g. Polar lock)

Place mode

Place and Spin mode
Polar Grid Utilities

- **Polar routing**
  - Special mode for drafting or routing in a circular or radial fashion
  - Snap to fine or coarse grid
  - Ability to follow radial and angular contour
  - Alternate path options (e.g. *Arc smart*)
Polar Grid Utilities

- Polar shape and void editing
  - Special mode for creating shapes and voids in a polar grid
  - Snap to fine or coarse grid
  - Ability to follow radial and angular contour
  - Alternate path options (e.g. **Arc smart**)

![Polar Grid Utilities Diagram]
Post Processing

• Central cockpit from which manufacturing output can be configured and generated

• **Hit Button** solution many users are asking for

• Tasks can be enabled / disabled, added and removed individually
  – Standard functions from PCB Editor
  – Skill procedures (e.g. FloWare)
  – User defined scripts

• Jobs can be defined which contain a collection of tasks

• Configuration stored in database

• Import and Export capabilities for the purpose of standardization
Post Processing

- Context menu provides task editing
  - Info
  - Tasks can be added (appended or inserted)
    - User tasks
    - Predefined tasks including arguments
  - Tasks can be removed if not needed at all
Post Processing

• Additional built-in functions
  – **Clean Data**
    Performs file delete operations based on arguments provided including wildcards
  – **Standard BOM**
    Generates standard BOM in text and csv format
  – **Standard Plot**
    Plots all artwork film control records into one PDF file artwork.pdf
  – **Rename Files**
    Provides powerful automated file renaming mechanism (wildcards, variables and pattern tags) for PLM purposes
Push to Grid

- Placement application featuring
  - Highlight off grid symbols
  - Option to move / push symbols to the nearest grid point
  - Regular placement mode for standard placement operations
- Single and multiple selected symbols
- Including mirror and spin support
Quick Symbol Edit

- Opens symbol editor out of a layout database for viewing or editing purposes

- Three modes
  - **Edit same session**
    - Acts similar to **Tools > Modify Design Padstack** (next slide)
  - **Edit new session**
    - Launches a new session and open the symbol dra file
  - **Export only**
    - Only exports symbol data (dra, psm, pad, etc.) for the selected symbol to a specified directory
Quick Symbol Edit

- **Edit same session**
  - Seamless edit and update of a symbol definition in board context
  - Acts like Tools > Modify Design Padstack
  - Steps:
    1. Launch **Quick Symbol Edit** in layout context
    2. Select a symbol
    3. Current database closes, symbol dra file will be opened
    4. Make your modifications
    5. Finally choose **File > Update board database**
    6. Specify update settings
    7. The layout database from which the command was launched opens again and the footprints will be updated
Replace Via

- More flexibility while replacing vias in the design
  - Specify padstack names
  - Restrict replacement to selection area
  - Include and exclude vias from the specified area through interactive commands
- Additional filter options, e.g. distinguish between
  - Vias with or without testpoint status
  - Mirrored or unmirrored vias
- Processing options
  - Allow DRC
  - Ignore Fixed
  - Retain Mirror Status
Shape Utilities

- Includes useful functions when editing shapes
  - Boolean operations
  - Size operations (Expand, Contract, Size)
  - Rectangular shapes with rounded corners
  - Advanced boundary edit functions
    - Stretch
    - Cut
    - Round off vertices
    - Change round off radius
    - Modify individual arc parameters
  - Modify shape priorities
  - Check status *(New)*
Shape Utilities – Boolean Operations

- **Features**
  - OR, AND, ANDNOT, XOR
  - Supports shape–shape operations as well as shape–line and shape–text operations
  - Shape attributes (shape type, fill style, net name etc.) are retained and applied to the resulting shape
  - Ability to split shapes across voids
  - Expand / contract capabilities
  - Useful for general editing and basic RF-applications

**Basic operations**

**Operation**
- Union (OR)
- Intersection (AND)
- Difference (ANDNOT)
- Symmetric difference (XOR)

**Shape processing**
- Expand/Contract
  - First: 0.000
  - Second: 0.000
- Include manual voids
- Split shape across voids (Useful on non-voidable subclasses, e.g. keepout)

**Result**
- Copy to class/subclass
- Etch
  - Bottom

Etching negative texts into copper

Cutting a dynamic shape
Shape Utilities – Boolean Operations

• Let’s you also split shapes across voids
• Useful when working on non-voidable layers such as KEEPOUT, NO_PROBE etc.
Shape Utilities – Size Operations

- Expand / Contract similar to Edit > ZCopy but directly acts on selected shape. No copies remain. Additional options for void handling

Expanding a shape

Scaling a shape
Shape Utilities – Rounded Rectangular Shapes

- Useful when defining pad shapes
- Size can be specified exactly
- Reference point can be chosen by using negative values for width and height

Basic operation

Specify shape origin
Shape Utilities – Advanced Boundary Edit

- Supports several modes
  - Stretching of shapes
  - Cutting of shapes
  - Rounding off vertices
  - Changing existing round off radius
  - Modifying individual arc segments of a shape

- All modes within one command
  - Including Oops support

- Additional features
  - Can also edit voids (Find Filter)
  - Supports single pick as well as window selection (several vertices)
  - Snap to grid, Trim to selection window
  - Display control (e.g. handle size)
Shape Utilities – Advanced Boundary Edit

• Use model
  1. Select the shape or void to be modified
  2. Select an action, e.g. **Round off vertex**
  3. Pick a vertex or drag a window to select the vertices
  4. Switch to other modes if necessary and continue
  5. RMB > Done
Shape Utilities – Advanced Boundary Edit

• Stretching of shapes
  – Specify offset if accurate stretch distance is needed

– Trim to selection window lets you control stretch point origin
Shape Utilities – Advanced Boundary Edit

• Cutting of shapes
  - Adjacent segments of selected vertices will be deleted

  ![User window](Image)

  - Trim to selection window simply cuts at the window boundary

  ![User window](Image)
Shape Utilities – Advanced Boundary Edit

• Round off vertex
Shape Utilities – Advanced Boundary Edit

• Change round off radius
  – Let’s you change existing rounded off corners

– Can also restore original vertices by specifying a radius of 0.0
Shape Utilities – Advanced Boundary Edit

• Change arc mode
  – Parameters of existing arcs can be changed in Options panel. End points of adjacent segments will be adjusted automatically. Tuning possible by using +/- buttons
Shape Utilities – Advanced Boundary Edit

- Other highlights
  - Stretch / Cut on arc segments in conjunction with Trim option
  - Round Off with adjacent arc segments
Shape Utilities – Priority

- Interactive application to modify the priority of a dynamic shape

- Use model
  - Select shapes by subsequent picks
  - Choose **RMB > Complete selection** to enter priority edit mode
  - Use one of the options and select a shape to modify the priority
    - Highest
    - Increase one level
    - Decrease one level
    - Lowest

- Priority numbers are displayed dynamically
Shape Utilities – Check Status

- Utility to check and highlight island and dangling shapes
  - Islands: 0 connections
  - Dangling shapes: Only 1 connection
- Such shapes can act as antennas and cause EMI issues
Shield Generator

- Facilitates the generation of shape and via pattern for shielding purposes
- Includes shield rings along board outline (e.g. for ESD protection) as well as the generation of shield boxes for RF circuits
Shield Generator

- Different modes for shield generation
  - Select Boundary
  - Draw Rectangle
  - Place Rectangle
  - Derive from Line
  - …

- Shape parameters
  - Direction, offset, width, layers, …

- Via parameters
  - Via gap, offset inside ring, initial offset, …

- Mask generation and cutting capabilities for solder mask and paste mask

- Ability to create groups or symbols
Shield Generator

• Derive from Line
  – This mode allows you to use construction lines as template and derive shield structure. Useful if shield box has partitions with arbitrary segmentations
Shield Routing

• Creates shield for critical signals in a semiautomatic way
  – Through interactive routing including dynamic preview
  – By selecting existing clines

• Support for side and tandem shields including parameters for gap and width and net name

• Support for dynamic or static shapes as well as route keepouts

• Advanced parameters can be used to control layers individually

• Parameters stored in database for each shield structure separately
  – Existing shields can be updated at any time without reentering information
Shield Routing

• Default shield parameters
  – For regular applications
  – Side shielding (width and gap)
  – Tandem shielding (width) on adjacent layers only
  – Shape style and net name

• Advanced parameters
  – Can be used to define parameters for several layers individually
  – Export / Import settings for reuse purposes
Shield Routing

- Routing shields interactively

Dynamic preview while routing with cursor

Final structure once snapping to pin or via
Shield Routing

- Updating existing shield structures due to routing modifications
  - Seamless update since parameters are stored in the database for each shield structure
Shield Routing

- Creating combined shields for several contiguous clines
  - Through **Temp Group** command

- Parameter probe
  - Simply copy parameters from one structure to another
  - Parameters will be extracted and can be applied to other clines
Shield Routing

- Via pattern generation
  - Use Advanced Parameters form to specify
  - Multiple rings can be specified
  - Settings which apply to padstacks, offsets, alignment, angles, etc.
Shield Routing

- Advanced via pattern settings
  - Multiple padstacks (e.g. microvias)
  - Parameter for main stitching gap and gaps within in the stack
  - Staggering option
Shield Routing

• Microvia stacking within ring

Contour

Outwards

Inwards

45 deg inwards
Silkscreen

- Powerful silkscreen utility
- Configuration
  Objects to silk e.g. lines and labels can be configured as well as obstacles which require silk objects to be cut e.g. soldermask, keepout areas
- Rules
  Clearances and minimum segment length can be defined
Silkscreen

- DRC
Before generating silkscreen data, a DRC check can be performed, indicating all violations helping users to identify problems and fix them before actual data is generated.
Snap Generator

- Creates snap points for standard PCB Editor commands
- Graphical entity in PCB Editor
- Snap point types
  - Intersection, extended intersection
  - Endpoints, center points, origins
  - Subdivision points (e.g. divide circle by 5)
- Stored in database
- Many options

Runs also when working on symbol drawings
Snap Generator

Snap point types

Placing component R5 to intersection I4 using Edit > Move
SVG Export

- Generates SVG data out of **PCB Editor**
- Export SVG from current drawing
- Export SVGs for a complete footprint library including HTM report generation
- Profile support
  - Content (layers) and styles (e.g. colors, opacity, non-vectorized texts, etc.) can be specified using predefined profiles
SVG Export

- Example for HTML library report
Synchronize Testprep

- Automates the testpoint assignment in PCB Editor when dummy testpoint symbols (one-pin components with Refdes TP*) have been used in the schematic
- Generates testpoint on corresponding pin in PCB
- Postprocessing can then be done using PCB Editor Testprep functionality
Variant Assembly

• Gives customers even more flexibility when creating variant assembly views

• Features
  – Automatic generation of all assembly variant views in one step
  – Customizable label content, e.g. Refdes, Value, Part_Number with the ability to annotate two labels per component
  – Customizable component outline, e.g. Assembly, Place_Bound, Silkscreen, …
  – Customizable style for DNI components, e.g. remove all or draw a thick cross through the label, …
  – Customizable style for Alternate components by using label prefix / suffix, changing colors or hatching component outline
  – Supports rule based coloring of objects, e.g. all SMT in green
  – Automatic mirroring of bottom view for better readability
  – Settings stored in database
Variant Assembly

User interface
Variant Assembly

Customizable label content to be displayed (default is Refdes)

Customizable component outline to be displayed (default is Assembly)
Variant Assembly

Available styles for DNI parts

Style „Remove All“

Style „Remove Label“

Style „Cross Graphics“

Style „Cross Label“
Module: Variant Assembly

Available styles for alternate parts

Style „Modify Labels“

Style „Modify Colors“

Style „Modify Graphics“
Module: Variant Assembly

Rule based coloring of components

- Color rules can be linked to individual properties
- Several match pattern per rule
- Wildcard support
Variant Assembly

Automatic mirroring of bottom view e.g. side by side to top view
Variant BOM

• Creates various reports (BOM) from layout database taking variant information into account
  – Generates reports (e.g. pick & place) data for a variant
  – Can be used for reports on core design also

• Configurable content
  – Any database attribute
  – Ability to distinguish between common and variant parts
  – Exclude BOM non-relevant parts (e.g. dummy testpoints)
  – Header information, order of columns, column width
  – Hierarchical sorting up to three levels
  – Option to split output files
  – Settings stored in database, export and import capabilities

• Output format
  – HTML, ASCII, CSV
Variant BOM

User Interface
Variant BOM

Report Formats

CSV

HTML

Text
Variant BOM

- Hierarchical sorting
  - Up to three levels
  - Options to add newline for level 1 and level 2
  - Options to split output files for level 1 and level 2
Variant BOM

- **Transformation rules**
  - Sometimes users need to transform x, y and rotation due to Pick & Place requirements from manufactory
  - E.g. always subtract 360 from angles given by PCB Editor for components on BOTTOM side
  - Formulas can be specified for TOP and BOTTOM separately
  - Unit specifiers such as MM, MILS etc. are supported
Z-DRC

- Checks clearances along the Z-axis
- Useful for applications which have to meet safety and explosion requirements
- Objects to be checked can be specified on net or netclass basis in any combination
- Spacing value can be specified manually or derived from DRC system (Constraint Manager)
- Graphical DRC: Markers & Overlapping
- Cross-probing and filter capabilities
Z-DRC

- The negative margins (amount of overlapping) can be displayed once a DRC has been selected in the Output tab
FloWare Modules
Schematic Entry

OrCAD Capture / Allegro Design Entry CIS
FloWare Modules Schematic Entry

- Flat Net Utilities
- Smart Aliases
- Split Wire
- Testpoint Check
- Tortoise SVN Integration
- Update Titleblock
Flat Net Utilities

• TCL applications with features to report and display the complete physical net (flat net) over all pages
• Very useful for review purposes when working with hierarchical designs
• Fully integrated into standard operations (e.g. select)
• Features
  – Report the physical net name in the command window while selecting wires in the canvas
  – Display the physical net name in tooltip
  – RMB > Highlight / De-highlight physical net over the full hierarchy
Flat Net Utilities

Physical net name report

Capture> Physical net(s) of selected wire(s): AD_IRQ2
Flat Net Utilities

Tooltip reporting physical net name
Flat Net Utilities

- Highlight / De-highlight physical net on all pages / hierarchies
**Smart Aliases**

- TCL application with smart editing and highlight capabilities when working with user defined signal names (aliases)

- Features
  - Dynamic highlighting of unnamed wires
  - Dynamic highlighting of wires with multiple different signal names
  - Smart rename when editing a signal name
Smart Aliases

• Dynamic Highlight

Different signal names on same "wire"

Wire without signal names
Smart Aliases

- **Smart Rename**: Keeps signal names in sync even if signal names have been assigned at multiple locations

  e.g. Rename MY_OUT to OUT1
Smart Aliases

- **Smart Rename**: Even helps resolving conflicts when different signals names were present

  e.g. Rename MY_OUT to OUT1
Split Wire

- Provides automatic wire split when components (e.g. resistors) are placed / dropped on wires
- Enable / Disable through context menu
Testpoint Check

• Check testpoint coverage for whole design

• Based on user-defined testpoint components (e.g. One-pin components TP*)

• Configurable search criteria
  – Reference = TP*
  – Value = Testpoint

• Status report
Tortoise SVN Integration

- **Features**
  - Import a project to a SVN directory
  - Committing your project to the server
  - Load the latest project version from the server
  - View the log-file
  - Checkout a directory
  - Lock / unlock a project
  - Compare – only with CIS license
  - Each command is directly available in Capture
  - This module always use your entire project directory

- **Prerequisites**
  - Tortoise SVN (Open Source)
  - SVN Server (Open Source)
Tortoise SVN Integration

Use model: Creating a new project

Developer 1
New Project
Import*

Repository

Developer 1
Project 1
Checkout*

Developer 2
Project 1
Checkout*

Stored on your personal drive

Stored on the SVN Server

* Commands launched from the Capture TortoiseSVN interface
Tortoise SVN Integration

Use model: During the daily use

* Commands launched from the Capture TortoiseSVN interface
Tortoise SVN Integration

- When committing a project, you can add messages which will be stored on your server, for example:
Tortoise SVN Integration

• View the log messages on your project history
Tortoise SVN Integration

• **Design Compare:**
The latest version from the SVN Server will be compared with your local version

![Design Compare screenshot](image)

**Local version** | **Version from the SVN Server**
--- | ---
Objects from E/TEMP/TESTDIFF/PROJECT1.DSN | Objects from E/TEMP/TESTDIFF/PROJECT1.DSN
  - SCHEMATIC1
    - TitleBlock
    - U1
    - U2
    - U7
  - COMP_INSTANCE | Design View
  - SCHEMATIC_VIEW
  - SCHEMATIC_PAGE
  - Schematic_Title_Block
  - U1
  - U2
  - U7
  - COMP_INSTANCE
  - NOT_PRESENT

• This command works only when you have a CIS license
Update Titleblock

- Automatic update of titleblock properties
- Driven by external configuration file customvar.cfg
- Variant aware

![Customvar.cfg Example](image)
Contact us / Kontakt zu FlowCAD

Please do not hesitate to contact us.
Für weitere Fragen und Informationen stehen wir gerne zur Verfügung.

FlowCAD Deutschland
Mozartstr. 2
85622 Feldkirchen bei München
T +49 89 45637-770
info@FlowCAD.de

FlowCAD Schweiz
Hintermättlistr. 1
5506 Mägenwil
T +41 56 485 91 91
info@FlowCAD.ch

FlowCAD Polska
ul. Sąsiedzka 2A
80-298 Gdańsk
T +48 58 727 90 90
info@FlowCAD.pl
International Contacts (1/2)

• For United Kingdom contact Parallel Systems
  – Pricing information, demo keys:
    • E-mail: sales@parallel-systems.co.uk  Phone: +44 1483 591 310
    • E-mail: info@parallel-systems.co.uk  Phone: +44 1483 591 310
  – Technical issues:
    • E-mail: support@parallel-systems.co.uk  Phone: +44 1483 591 31

• For Denmark and Iceland contact Nordcad Systems A/S
  – Pricing and info: info@nordcad.dk  Phone: +45 96 31 56 90
  – Support: support@nordcad.dk  Phone: +45 96 31 56 99

• For France contact Artedas France
  – Pricing and info: info@artedas.fr  Phone: +33 170916720
  – Support: techsup@artedas.fr
International Contacts (2/2)

- For Benelux contact CB Distribution BV
  - Pricing and info: info@cb-distribution.nl  Phone: +31 74 256 1424
  - Support: support@cb-distribution.nl  Phone: +31 74 256 1424

- For Spain and Portugal contact CB Distribution Southern Europe SL
  - Pricing and info: info@cb-distribution.es  Phone: +34 91 002 4576
  - Support: support@cb-distribution.es  Phone: +34 91 002 4576

- For North America contact EMA
  - Pricing and info: info@ema-eda.com  Phone: +1 800 813 7494
  - Support: techsupport@ema-eda.com  Phone: +1 585 334 6001
Follow Us

FlowCAD.com/newsletter
The FlowCAD newsletter for PCB designers appears about every two months. It is free of cost and will be sent by e-mail.

youtube.com/FlowCAD
On our YouTube channel you can find 100+ tutorials to learn more about electronic circuits. Our playlists also offer product news and webinars.

twitter.com/FlowCAD
On FlowCAD’s Twitter we provide press releases, news articles, films and images as well as reports from events.

facebook.com/FlowCAD
Join our Facebook page. You will find selected news, events, success stories and insights.

Don’t forget to subscribe, share and like!