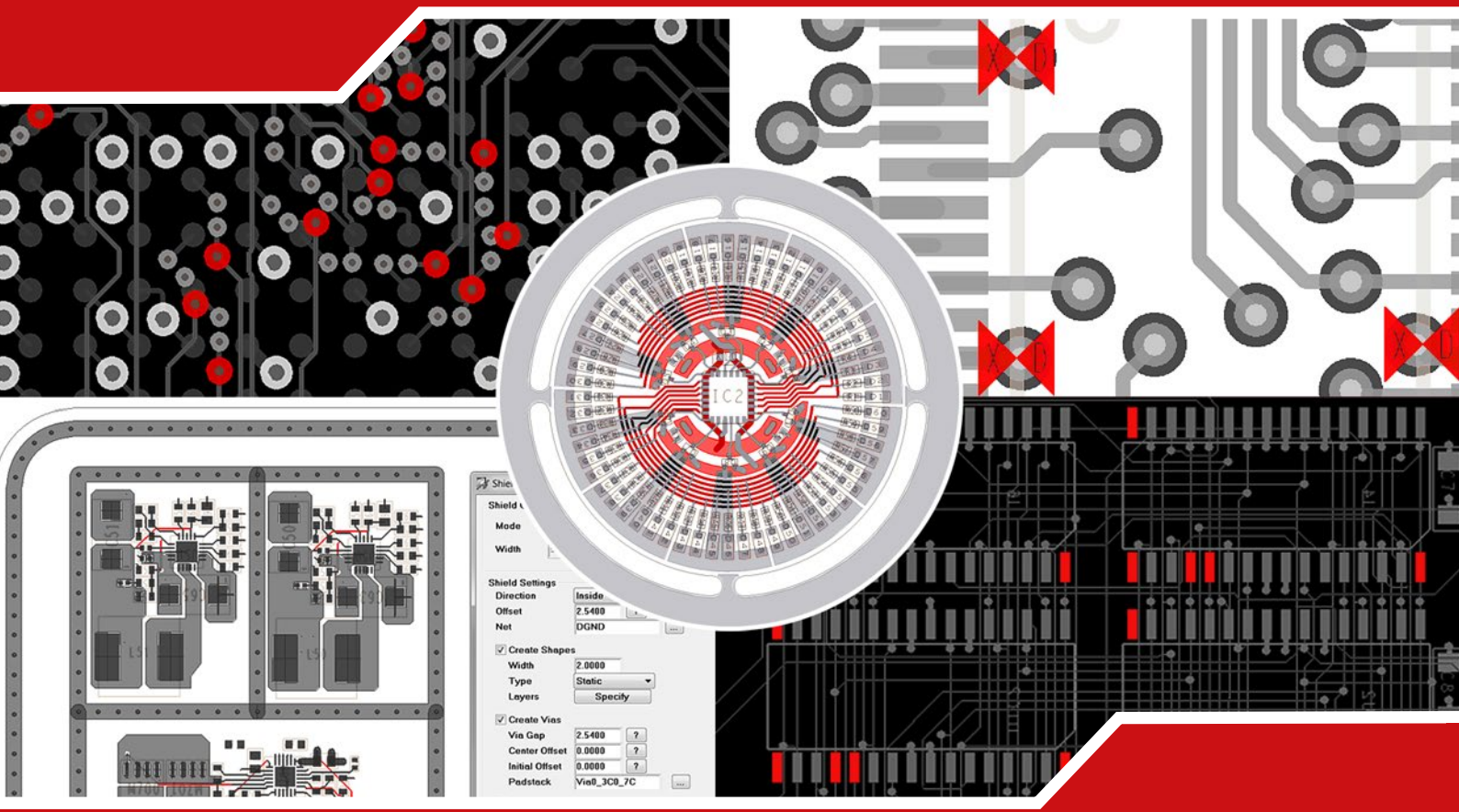
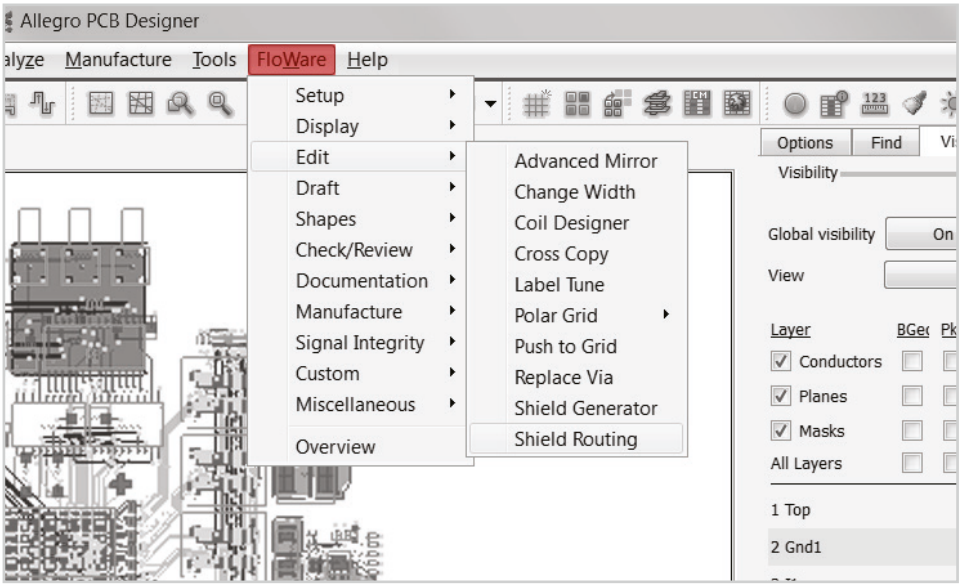
FlowCAD

FloWare Modules



Product Description

FlowCAD



Integrated menu in OrCAD/Allegro PCB Editor for FloWare Modules

FloWare Modules

PCB Editor from OrCAD and Allegro includes a programming language (SKILL) which makes it possible to program customized add-ons for clients. FlowCAD offers a selection of functions as FloWare modules which were created to meet the specific needs of our customers.

If you wish to integrate additional functions into PCB Editor yourself using these open programming interfaces, SKILL programming courses are offered at the Cadence Training Center. If you prefer, the FlowCAD team can program additional functions to meet your specific needs for you.

Installation of FloWare modules is a simple process and does not require any expert knowledge. There is an installer, which copies the required files into the right folders. The installation process includes a wizard and guides you through the FloWare installation process.

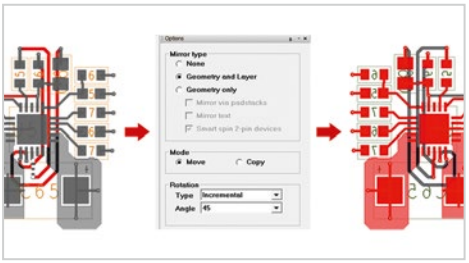
The menu structures are automatically recognized by PCB Editor and shown in the tool bar.

All FloWare modules are downloadable and the license is distributed by e-mail. The license is valid for the site of your company location. A detailed PDF file documents the functionality and is included with delivery.

The following modules are currently available:

- Advanced Mirror
- Advanced Testpoint Check
- Anti Tamper Mesh PCB
- AOI Check
- Assign Net to Via (Change Net)
- Barcode Generator
- Batch Plot
- CAF-DRC
- Change Width
- Class Color
- Cleanliness Check
- Coil Designer
- Contour Place
- Cross Copy
- Cross Section Generator
- Custom Variables
- Design Compare
- Drafting Utilities
- Drawing Designer
- Drawing Size
- Drawing View Manager
- Edge Plating
- FPGA Utilities
- Highlight Dummy Pins
- IBIS Prototype Modeler
- Label Generator
- Label Tune
- Mask Generator
- NC Panel Route
- 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 3D
- Variant Assembly
- Variant BOM
- Z-DRC

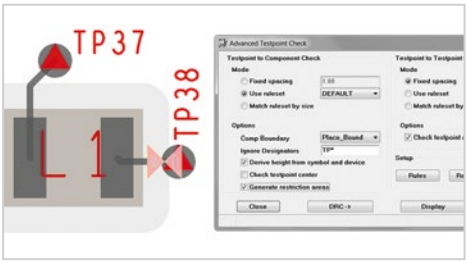
FloWare Add-Ons



Enables mirror operations

Advanced Mirror

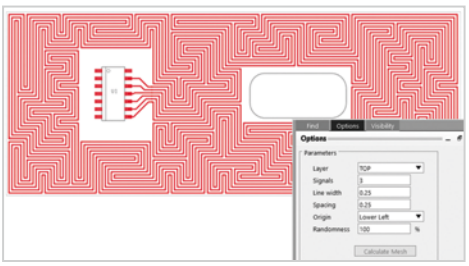
Advanced Mirror is an application that allows users to perform mirror operations while moving or copying a group of selected objects. Mirroring can be performed either across subclasses or on the same subclass (Geometry only). When wires and components are mirrored, DRCs will occur because the component pins no longer match the mirrored layout. Many users consider this new connection of the pins to be much easier than rebuilding the complete circuit board. It is often used to mirror keepouts and shapes.



Addresses various rules for testpoint checking

Advanced Testpoint Check

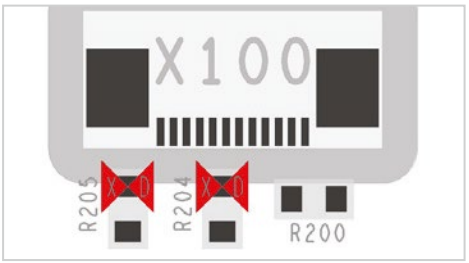
In order to optimally fulfill contacting requirements for In-circuit tests (ICT) various test probes are available. These differ in terms of height, size, tip style and type of connection. On the PCB the corresponding contact areas (testpoints) must reflect these requirements. Advanced Testpoint Check is a toolbox application which addresses various rules for testpoint checking.



Convoluted maze of wires (mesh), interactively created

Anti Tamper Mesh PCB

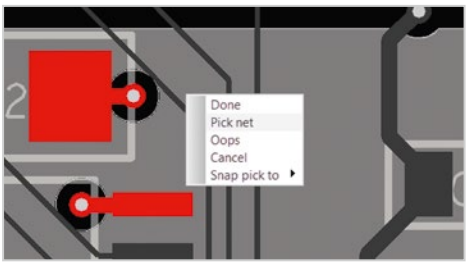
Anti Tamper Mesh PCB interactively creates meshes, which have the main purpose to actively detect and respond to external attacks by monitoring changes in resistance, capacitance, breaks or shorts. Convoluted mesh of wires on the outer layers of the PCB are often required for Hardware Security Modules (HSM). After specifying the mesh region and definition of keep outs, the starting edge needs to be selected, from which the signals start and end. Finally the mesh is calculated.



Helps users to check AOI related rules

AOI Check

This FloWare module helps users to check AOI related rules directly in PCB Editor. Shadowing can cause serious issues in verification process. Shadows can be calculated in various directions based on specified camera angles (all, horizontal, vertical, 45 degrees) taking component height into account. Special rules apply to 3D inspection systems.



Assign Net to Via

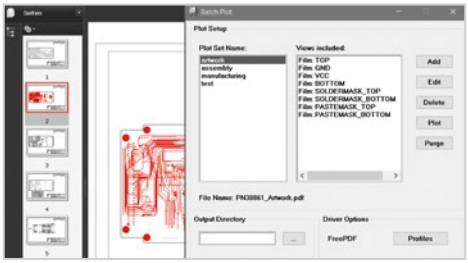
Assign Net to Via

While working on a PCB layout in OrCAD PCB Editor or Allegro PCB Editor it happens, that you place a via with no connection to a net. There are several reasons for this scenario, maybe you want to use the via as a testpoint and the placement is predefined by the already existing test adapter or you have other reasons for the placement driven from the mechanical design. With the FloWare module Assign Net to Via you can assign easily any net to the stand alone via. If you now start routing from the via, it already has the net information and will respect assigned constraints.

Barcodes in PCB Editor

Barcode Generator

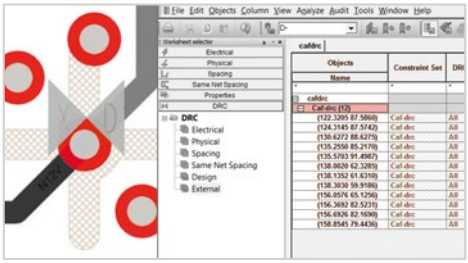
To identify PCBs some customers print barcodes on a PCB layer as part of the manufacturing process. With this FloWare module you can easily enter the value and create the barcode of that value. The barcode is placed onto any layer of the design. The app supports symbol definitions for Code 39, Code 128, QR Code and DataMatrix with adjustable parameters for barcode height, width, single bar width and margins. You have the choice, if you want to show the text under the bars or choose an inverted display. For easy dimensioning you have a dynamic preview during parameter change.



Defining output data as a batch file

Batch Plot

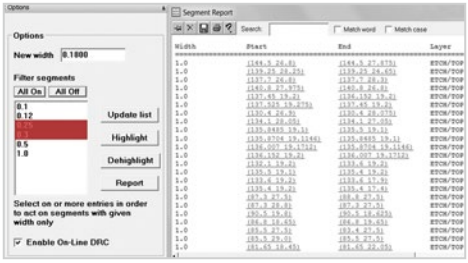
In order to automate production data output (Gerber data, drill plans, …) it is advisable to create a defined document batch. The FloWare module Batch Plot supports this function. The user can create a whole batch of data in one step as a PDF file. You write the data using a PDF print driver (e.g. Adobe Acrobat) in one or more multi-page documents. The order of the pages in the data batch can be defined by the user. The settings for complete document batches for production, assembly and tests are saved.



Special DRC for CAF on PCBs

CAF-DRC

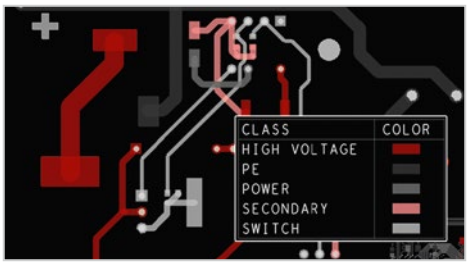
CAF (Conductive Anodic Filament) describes the chemical effect of copper ion migration in the FR-4 base material at high voltages, which causes breakdowns in the printed circuit board. The FloWare module CAF-DRC makes it possible to execute a special design Rule Check in the PCB Editor for OrCAD and Allegro. The corresponding minimum distances between the bore outer diameter and the next conductive material are specified as a function of voltage classes. If the distance is undercut, a DRC error occurs.



Change the width of clines and clines segments

Change Width

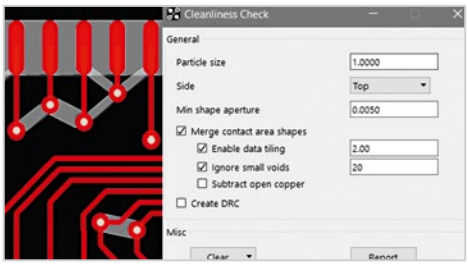
Change Width helps users to change the width of clines and clines segments. In contrast to the standard "Edit – Change" command this module supports a filter mechanism in that the changes are only applied to segments matching a given width. Furthermore highlight and report functionality is available. Users can select by pick, window, temp group or find by name.



Nets colored by class

Class Color

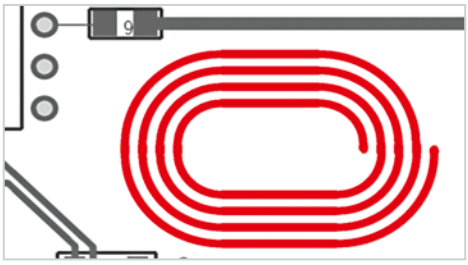
With this FloWare module for OrCAD and Allegro PCB Editor you can colorize different nets. This is helpful, when you have high voltages on the PCB and you have to ensure a certain net spacing between several net classes for isolation. When coloring each voltage range in one color, you can visually review your design rules and identify missing or wrong design rule settings. This feature helps you when reviewing a design or need to document spacing classes.



Cleanliness Check calculates and visualizes contact areas

Cleanliness Check

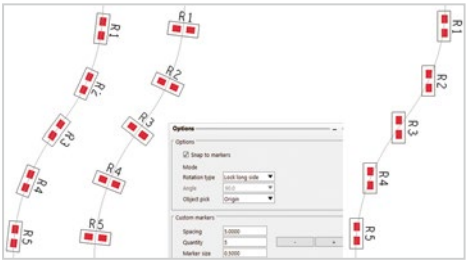
Cleanliness Check analyzes the PCB layout for contact areas caused by particles with a given size. Contact area specifies the area in which a particle will cause a short, no matter how the particle is oriented. Cleanliness Check calculates and visualizes contact areas using shapes and writes a report. By sweeping the particle size the information from the output can be used to feed cleanliness assessment calculator from ZVEI.



Generate planar PCB windings

Coil Designer

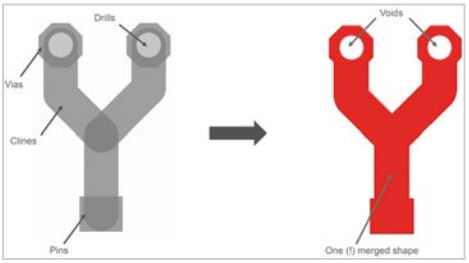
With FloWare module Coil Designer you can quickly create planar windings for transformers in PCB Editor. The generation is controlled with variable parameters. Available are four general shapes: round, rectangular, hexagonal and octagonal. With changes in the parameters in the menu the dimensions or number of windings will be dynamically updated in the preview at the cursor. Corners can be mitered or rounded off with a parameterized radius. Within a structure a keep out area can be generated as well with a spacing towards the inner winding. If you want to place a via at the beginning and end of the winding, you can select an available via type. Winding direction can be clockwise or counterclockwise and the geometry can be rotated.



Examples with various alignment

Contour Place

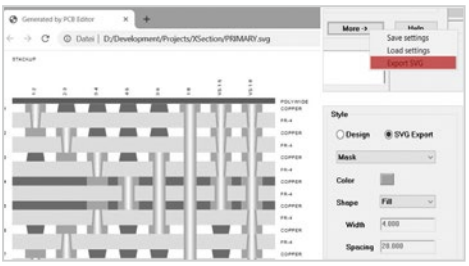
Contour Place lets you place components along a given contour. Compared to Polar Grid, it provides a more generic approach as arbitrary paths are supported along which components can be placed. Contour path is generated from existing objects. This module offers interactive placement of components, with automatic snap to contour path. Absolute or relative angles, lock long side or lock short side offers various alignment capabilities. In some cases, components need to be placed at certain, predefined locations on the contour path. For this purpose, custom snap markers can be generated.



Merging data to soldermask shape

Cross Copy

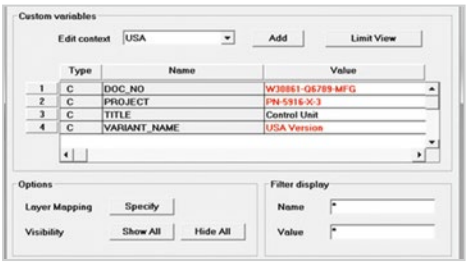
Some users wish to expand the copy function of PCB Editor. Using the command Edit-Copy in Cross Copy, a target layer can be defined. Standard copy command does not support a destination layer and compared to the Z-Copy command, Cross Copy offers way more flexibility.



SVG Export, e.g. for documentation purposes

Cross Section Generator

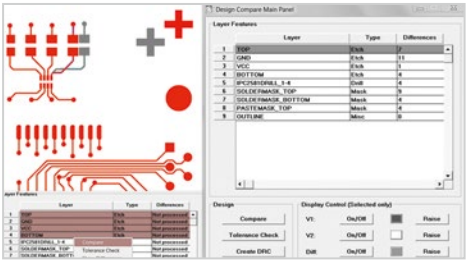
For documentation purpose the cross section (PCB stack up) needs to be documented. This module creates a documentation view of the cross section which is stored in the PCB Editor database. The appearance is customized for the content, which might include: layer name, layer thickness (including total thickness), layer material, via stack and via labels. Various graphical options can be set up like symbol size (extra row column spacing), fill styles for conductor and dielectrica, scaling of layer thickness for better readability. The configuration of the appearance is stored in the database and an updated plot will be generated without entering all settings.



Managing custom variables like date, user, ...

Custom Variables

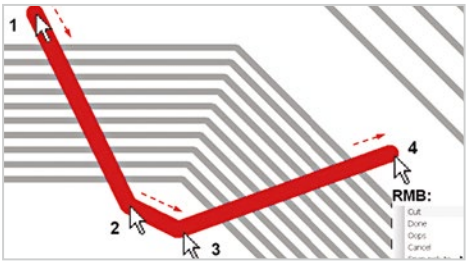
In PCB Editor the user can define his own variables to be used in the title block or as etch text. The variables are automatically updated across all subclasses and a placeholder can reference to more than one variable. The placeholder attributes (x, y, block, rotation) are stored in the database. The values for the variables can be entered manually, sourced from an external control file or even a cpm-file. Special fields are available for inserting Auto_Date, Auto_User, and others.



Useful when tracking changes in the product lifecycle

Design Compare

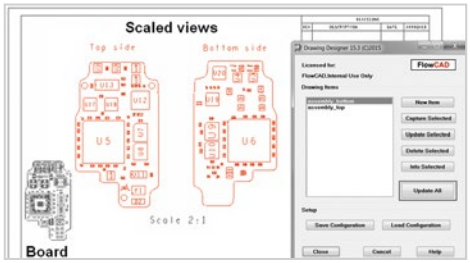
Design Compare compares two design databases and recognizes the differences. In this way deviations in the product life cycle can be detected and documented. The application distinguishes designs in two different types of comparison. In the standard comparison, parameters in the design that exist as numbers or texts (e.g., net names, properties) in the database are compared. In addition, a graphical comparison of the geometry of the layout is possible. The results can be output as an HTML report.



Cutting a bus in three segments

Drafting Utilities

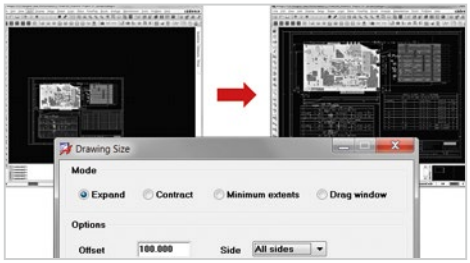
When you need to manipulate geometric forms you normally can do the operations much easier in a mechanical CAD software compared to OrCAD PCB Editor or Allegro PCB Editor. But FloWare module Drafting Utilities offers some useful functions to edit geometric PCB forms. It is possible to cut through forms and structures and use the separated individual pieces. This helps to create unusual shapes which are not based on a grid but on intersections of other objects. Besides the cutting feature there are some useful drawing features for lines and arcs.



Drawing Details for Manufacturing

Drawing Designer

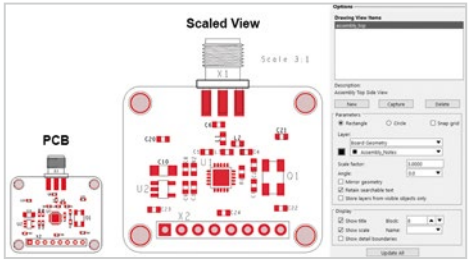
Manufacturing departments often need a zoomed area of a PCB for a detailed view of a special area. In PCB Editor you can create such a drawing detail. But FloWare module Drawing Designer offers more functionality. Details can be created on an additional documentation layer with a defined zoom factor and are always in synchronization with the electrical layers of the base design. Created details can be rotated or mirrored on documentations layers and the synchronization is done automatically. Settings are stored in the design database and are available when the design was opened. Settings can also be exported for usage in other designs.



Quick and easy way to change the design extents

Drawing Size

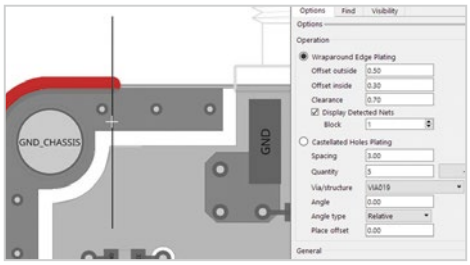
The FloWare module Drawing Size allows users to change and modify the drawing extents in a quick and easy way. This is useful if you want to reduce design extents to minimum values. Settings can be applied to all or individual sides.



Easy creation of manufacturing drawings

Drawing View Manager

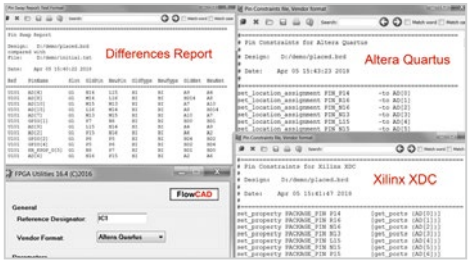
When releasing a PCB design for production, additional documentation is required for manufacturing, assembly or testing. Drawing View Manager facilitates the creation of manufacturing drawings. Scale factor or mirrored geometry, rotation and other parameters can be specified. Changes to the master PCB can be applied with a single click. User-defined templates simplify the work.



Wraparound Edge Plating

Edge Plating

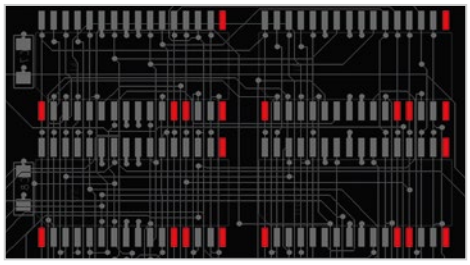
Edge plating and castellation used metallization at the sides of a PCB. It is used for EMI-shielding, thermal heat distribution as part of a cooling solution, better current distribution for power electronics, mechanical protection when the PCB slides in a ground connector or to solder a PCB directly on top of another PCB without a connector. The app supports two modes of operation, wraparound edge plating and castellated holes.



Reports for FPGA Flow

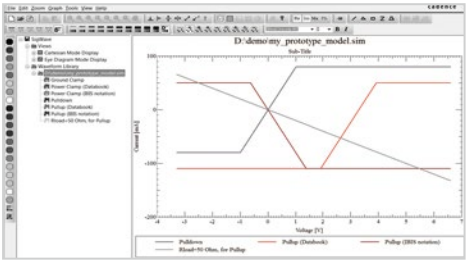
FPGA Utilities

When routing FGPAs on a circuit board, it is often necessary to swap individual pins (pin swap) to optimize routing. These changes must be saved in the FPGA environment so that the FPGA designer can synchronize his data. FPGA Utilities generates various reports for this purpose (e.g. Show Difference). Pin Constraints in manufacturer-specific formats (MicroSemi / Actel, Xilinx, Altera, Lattice or Excel / CSV) can also be generated which the FPGA tool can use directly. FPGA Utilities enables the user to synchronize FPGA and PCB design data quickly and efficiently.


Unconnected pins are highlighted

Highlight Dummy Pins

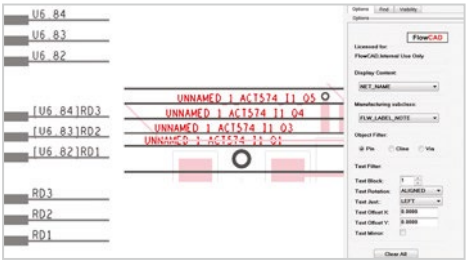
With FloWare module Highlight Dummy Pins you can assign a color to each unconnected pin. In a report you can see all unconnected pins and walk them through one by one. Cross probing from the report will zoom in the pin in the design. This app is useful for reviews of a design and you need to verify each unconnected pin, if it was unconnected intentionally or unintentional.



Creates scalable IBIS models for early stage SI analysis

IBIS Prototype Modeler

While designing and simulating a new PCB, there are situations where simulation models from component vendors are not available yet. For this reason users may want to start with a linear model first with a given voltage swing and an adjustable internal resistance based on information from datasheet. Creating an IBIS model from scratch using a text editor is time consuming and error prone. IBIS Prototype Modeler is a Floware module that allows users to create linear IBIS models on the fly for early stage simulation purposes in Sigxp. It offers all important features like the ability to create Push/Pull, Open-Sink and Open-Source buffers, adjustable voltage levels, etc.



Documentation of traces in the layout

Label Generator

In the documentation of circuit boards, it makes sense to name individual traces. With Label Generator text can be produced automatically through selection and stored on a layer for documentation purposes. Such text can then be hidden and unhidden. If necessary, text can be created from the available information in the database. It is possible to depict the pin name, signal name or combination of pin and signal name. The user positions text depending on selected objects and can make adjustments assisted by reference lines.

Adjustments to the labeling of components

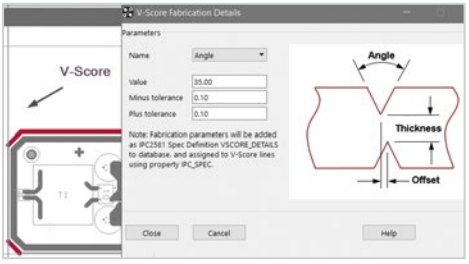
Label Tune

The positioning of reference labels in the library is normally standardized. After placement or in the case of high-density designs, texts can become difficult to read. Manual adjustment of text parameters is then necessary. With Label Tune, label size, rotation, and position can be adjusted so that they are easier to read for Assembly Drawings. Further parameters are mirror and center fit, which incorporate maximum block size and spacing to component boundaries.

Oversized solder mask

Mask Generator

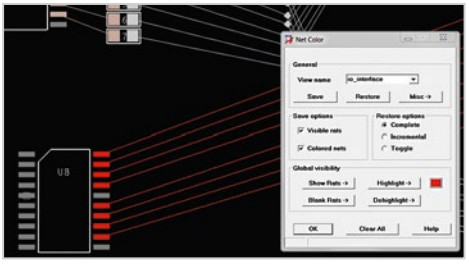
If you change your supplier for PCB fabrication, it might be necessary to change your masks sets. FloWare module Mask Generator offers additional features in OrCAD PCB Editor and Allegro PCB Editor to change (expand/contract) in a post process the mask data (i.e. solder mask and paste mask). The modified data will be stored on an additional documentation layer, so the base design stays unchanged.



V-Score Fabrication Details

NC Panel Route

This toolset supports various techniques for PCBs in a panel to be easily separated after they are manufactured and assembled. It is possible to add mill contour, split and cut mill and to add mill tabs with or without perforation drills. The app provides libraries from different PCB fabricators. V-Score lines can be defined in horizontal or vertical directions by selecting an appropriate side of the board. V-Score fabrication details can be specified which creates an IPC-2581 Spec Definition entry in the database.



Net Color View

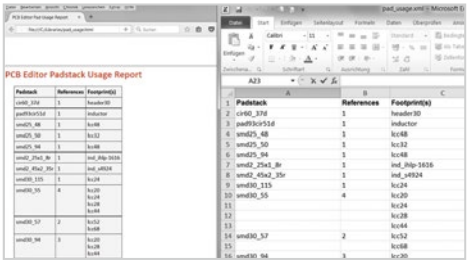
Net Color View

Net Color View allows saving and restoring net color and rat visibility settings. This is very useful for floorplanning and route feasibility studies. Temporary colored nets for a special task are easier to identify.

Different color per padstack

Padstack Finder

With FloWare Padstack Finder you can search in OrCAD PCB Editor or Allegro PCB Editor in the current design for pad stacks and highlight them. It is differentiated, if the pad stack is a pin or a via. The highlighted results are also available in a report and can be zoomed in by cross probing from the report. Different colors can be assigned to groups for highlighting. The filters Pad stack name and Drill diameter are available.

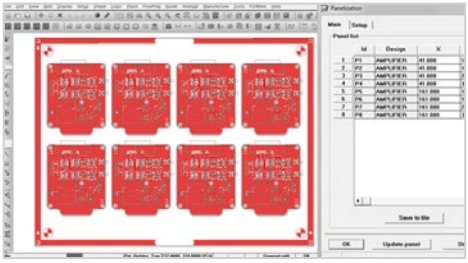


Report examples in formats HTML (left) and XML

Padstack Usage

Padstack Usage module allows users to generate padstack reports by extracting data from PCB Editor symbol libraries (*.dra). Different reports output formatted text, MS Open XML, and HTML:

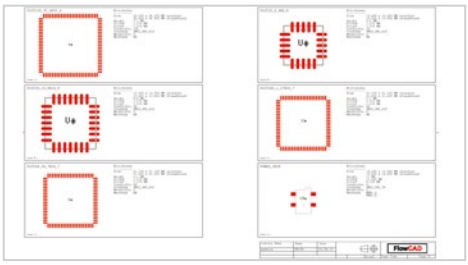
- Where-Used: A report which lists all footprints that use a given padstack.
- Padstack Standard: A report that lists the padstack definitions for a given footprint.
- Padstack Detailed: A report which lists detailed information for a given footprint. This information includes number of pins, vias and mechanical pins, pin numbers, xy coordinates etc.



PCB fabrication panel

Panelization

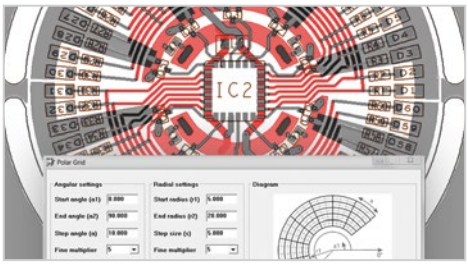
PCBs are manufactured often in fabrications panels. If the placement is not done by the external manufacturing company, the PCB designer can use the FloWare module Panelization to do it himself. Just open a new blank project in PCB Editor and place a manufacturing frame and fiducials. Now you can place routed designs multiple times in the panel. The design data will be linked with x and y coordinates and rotation information. While placing a design on the panel you can see the dimensions as a preview. Multiple placements of the design will automatically add a defined prefix to each instance of the design to avoid naming conflicts (duplicates).



Catalog of all footprints in a library

PCB Library Plot

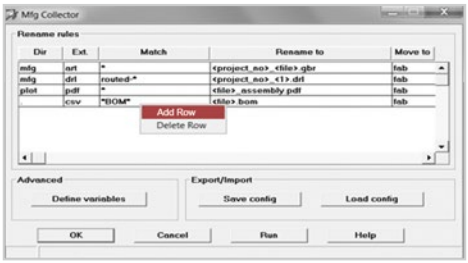
During development electrical engineers would like to see which PCB footprints are available in the company's library. This module will process a complete library and create a PDF document with a graphical representation of all available footprints. The graphics will be scaled to fit into a selected template (i.e. 4-up, 6-up, 8-up). In addition to the footprint attributes are added into each field, like footprint name, dimensions (place bound, visible), height, pitch, used pad stacks, pin count, mechanical pin count, etc. Which attributes are printed is customized. The add-on can run through different libraries.



Useful when circular placement and routing is required

Polar Grid

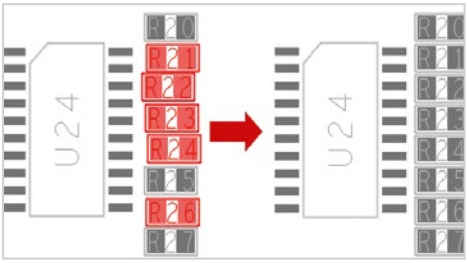
In some applications (e.g. medical, automotive) not only the boards have a circular outline also placement and routing has to be done in a circular fashion. Doing these kinds of boards on the basis of a cartesian grid is time consuming and cumbersome.
Polar Grid Utilities is a toolkit that is exactly dedicated to this kind of application. Besides the capability to define a polar grid, it offers additional functions such as polar placement, polar routing and polar shapes including voids.



Defining standard output jobs

Post Processing

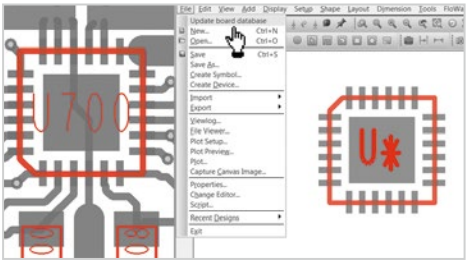
If a company always orders their PCBs from the same manufacturer, the required manufacturing data is also always the same. The FloWare module Post Processing offers a feature set to generate the output in always the same way in OrCAD and Allegro PCB Editor. You can generate sets of output data of existing reports in always the same order. Before you can run scripts or other SKILL (FloWare) routines to define parameter settings. The configuration will be saved in the database and the output job can be created with exactly the same settings. You can define in Post Processing different output jobs and start them individually. The created files can follow a naming convention (wild cards, variables and pattern tags).



Placement application

Push to Grid

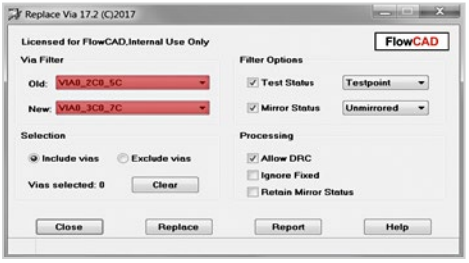
Push to Grid is a placement application that allows users to highlight off grid symbols including an option to move/push them to the nearest grid point. Furthermore it supports a regular placement mode for standard placement operations.



Shorten the Symbol Edit Flow in PCB Editor

Quick Symbol Edit

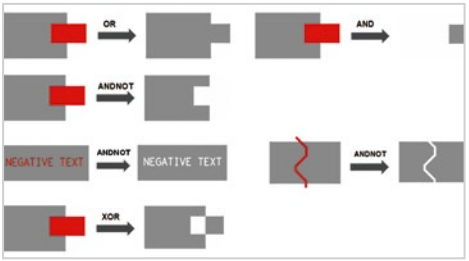
FloWare module Quick Symbol Edit was developed for such users who want to speed up the librarian work flow. There are three typical use cases for this app in OrCAD PCB Editor or Allegro PCB Editor. At first there is a possibility to change a symbol in the same session, which is similar to the Modify Design Padstack handling. The second methodology is to open a new session with a pre loaded Symbol.dra file. And finally you have the choice to export the symbol as .dra, .psm, .pad etc. in a specified directory. The file opening and closing will be handled by the app automatically.



Select the vias to be exchanged

Replace Via

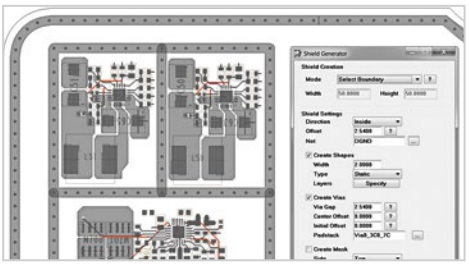
Replace Via is an application which gives users some more flexibility while replacing via padstacks in the design. The operation can be restricted to a selected area. Vias can be included and excluded through interactive commands taking various filter criteria into account. Additional processing options such as Ignore DRC or Retain Mirror Status are available.



Modification of shapes

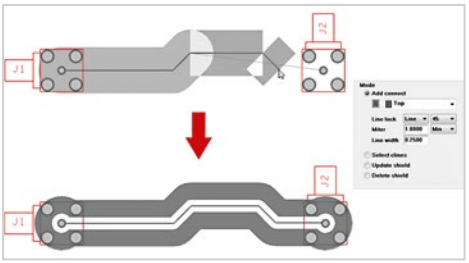
Shape Utilities

Copper shapes have to have sometimes very specific outlines for electrical reasons to enable a circuit board to perform correctly. FloWare app Shape Utilities can efficiently enhance the existing functionality in OrCAD PCB Editor or Allegro PCB Editor. With boolean operations (OR, AND, ANDNOT, XOR) two shapes can be calculated against each other to a new shape. The original shape attributes (shape type, fill style, net name, ...) of the primary shape will remain attached to the resulting shape. Any shape can also be scaled with size operators (Expand/Contract). A separate parameter will specify the handling of voids. Shapes with corners can be rounded off.


Easy shape and via pattern generation for shielding purposes

Shield Generator

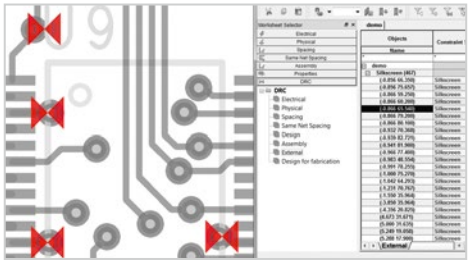
Shield Generator is a toolbox application which facilitates the generation of shape and via pattern for shielding purposes. This includes shield rings along board outline (e.g. for ESD protection) as well as generation of shield boxes for RF circuits which require additional noise reduction. The user can choose between different modes for shield generation and can set parameters for shape and via. Mask generation and cutting capabilities for solder mask and paste mask can also be used.



Creates shield for critical signals in a semiautomatic way

Shield Routing

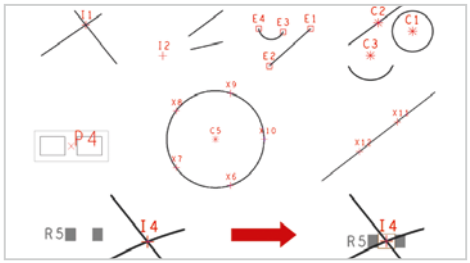
Some RF applications require shields for critical signals in order to minimize crosstalk and noise. Shields may be realized on the same layer like the signal trace (side shield) or on adjacent layers above or below the signal trace (tandem shield). In both cases the shield structure follows the structure of the signal trace and is expanded to a certain extent. Usually shields are realized by dynamic or static shapes connected to a ground net. Route keepouts may be also understood as some sort of shielding as they keep noise away. Shield Routing is an application which enables users to create shields for signal traces inside PCB Editor.



Silkscreen DRC

Silkscreen

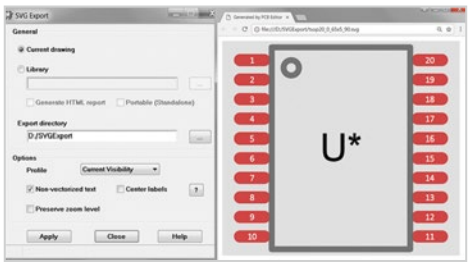
If printed circuit boards are densely populated, it may happen that there are vias or other elements under the silkscreen that should not be overprinted. With the module Silkscreen DRC design rules can be defined. Based on these rules, the module analyzes the PCB design and outputs corresponding errors according to the design rule check. The designer can decide if and how these bugs are eliminated. The module deletes according to the settings specifically silkscreen at the desired locations.



Snap points for geometrical adjustments

Snap Generator

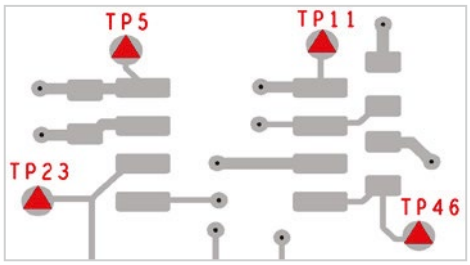
The graphical editing of copper areas, pads or other geometrical forms is sometimes complex and goes beyond the standard functions of PCB Editor. With Snap Generator it is possible to generate intersections of objects (I = intersection), center points (C), end points (E), pins (P = origin) or equally long sections (X = section) by selecting the elements. The generated snap points are saved on a separate layer and their coordinates can be combined with normal commands in PCB Editor. In this way, all special points outside the grid can be incorporated reliably and accurately.



Generates SVG data out of PCB Editor

SVG Export

SVG Export is an application which allows users to generate SVG data out of PCB Editor. Users can export SVG from current drawing, and for a complete footprint library (including HTM report generation). Content and styles can be specified by using predefined profiles. SVG is a vector file format for pictures, which is small in size and displays well on screens, and print as sharp image.



Test points synchronized with schematic

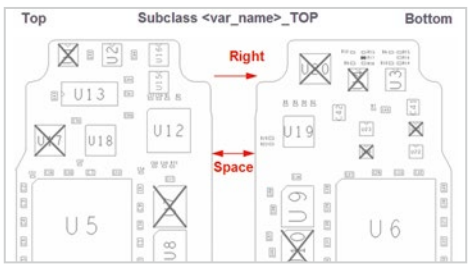
Synchronize Testprep

The FloWare module Synchronize Testprep can help in OrCAD PCB Editor or Allegro PCB Editor to assign a test point when using dummy test symbols (1-pin part with RefDes TP*) in the schematic. This app will generate from the schematic a corresponding test point in the PCB layout data base. Afterwards all PCB Editor post processing routines for test points (Testprep) can be used as usual. The menu offers various parameters to create a drawing detail. You can select the target layer, rotation, scale factor, and mirror. Pads can be filled or left unfilled and shapes can be displayed as they were defined in the design, unfilled, or hatched.

Multiboard panel

Variant 3D

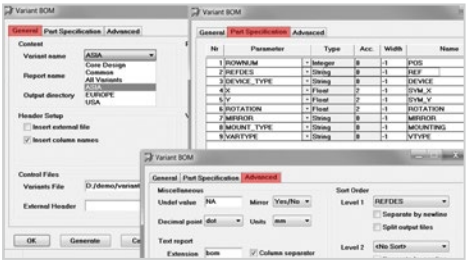
Variant 3D is a FloWare module to visualize variants in 3D canvas. Using the variant information from the design, DNI components will automatically be hidden in 3D. Single boards, regular panels and multiboard panels are supported.



Customized assembly drawings for variants

Variant Assembly

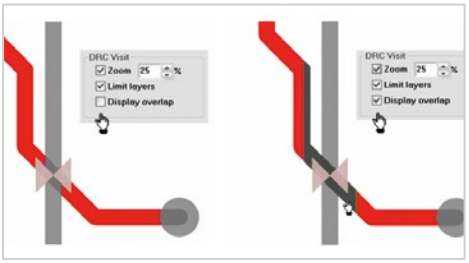
This module gives users more flexibility when creating variant assembly views. Production of variant views in one step is automated. Label content is customized (RefDes, Value, Part Number). The outline is also customized to assembly, place boundary, silk screen, or any other outline from the data base. Different styles for DNI (do not install) components are provided: remove all components, draw a thick cross through the label and many more styles. Alternate symbols can be indicated with a modified label with prefix or suffix or different component outline line style or hatched filling. In the output a automatic mirror for the bottom side can be placed.



Variant BOM user interface

Variant BOM

This add-on creates various BOM reports (Bill of Material) from PCB Editor database taking existing variant information into account. It can generate a pick and place report for each variant or the core design. The content is customized for any database attribute. Header information can be added as well as selecting the number of columns and their order.
All settings will be stored to easy regenerate the report after design modifications. The report can be generated in ASCII, HTML or CSV.



DRC for violations in Z-coordinates

Z-DRC

DRCs in PCB Editor work only in x and y direction. Z-DRC checks can be made respective the z-coordinates. This might be useful for safety or explosive requirements. The user can select between which layers the check shall be performed. As a result a list of Z-DRCs will be displayed. By clicking on each Z-DRC PCB Editor will zoom in on the area, where the error was found. The DRCs can be stored as external DRCs in the database.

Order Information

FloWare Add-Ons are sold in Central Europe by FlowCAD directly. Please contact your local FlowCAD office for a quotation. In the rest of the world FloWare is sold by other Cadence Channel Partners, who also sell the OrCAD and Allegro product line of Cadence's PCB solution. For more information please contact your local sales representative.

Licensing

FloWare apps are licensed per customer site (location). If a company has i.e. two R&D locations with 5 designers at each site, only 2 licenses per app need to be purchased. The software can be installed and used in parallel for all users at each location.
If you are interested in all apps, there are special bundle prices available as well for big corporations with many sites are global licenses available.

Installation

Installation of FloWare modules is a simple process and does not require any expert knowledge. There is an installer, which copies the required files into the right folders. The installation process includes a wizard and guides you through the FloWare installation process.

Videos on YouTube

Visit our **FloWare playlist** and have a look at some of the apps. You will also find there a top 10 video of the tools that increase your productivity in PCB design.

FloWare is available through your local Cadence Channel Partner:

Denmark
Nordcad Systems A/S

France
ARTEDAS FRANCE

Israel
EDA Integrity Solutions Ltd.

Italy
ARTEDAS ITALY

Netherlands
CB Distribution BV

Russia
PCB Soft

Sweden
Nordcad AB

Taiwan
Graser Technology Corp.

UK
Parallel System Limited

USA
EMA Design Automation, Inc.

FlowCAD

FlowCAD (Deutschland)

Mozartstr. 2
85622 Feldkirchen bei München
Germany

T +49 89 45 637-770
F +49 89 45 637-790

info@FlowCAD.de
www.FlowCAD.de

FlowCAD (Schweiz)

Hintermättlistr. 1
5506 Mägenwil (Aargau)
Switzerland

T +41 56 485 91 91
F +41 56 485 91 95

info@FlowCAD.ch
www.FlowCAD.ch

FlowCAD (Polska)

ul. Sąsiedzka 2A
80-298 Gdańsk
Poland

T +48 58 732 74 77
F +48 58 732 72 37

info@FlowCAD.pl
www.FlowCAD.pl