Quickstart OrCAD PCB Editor

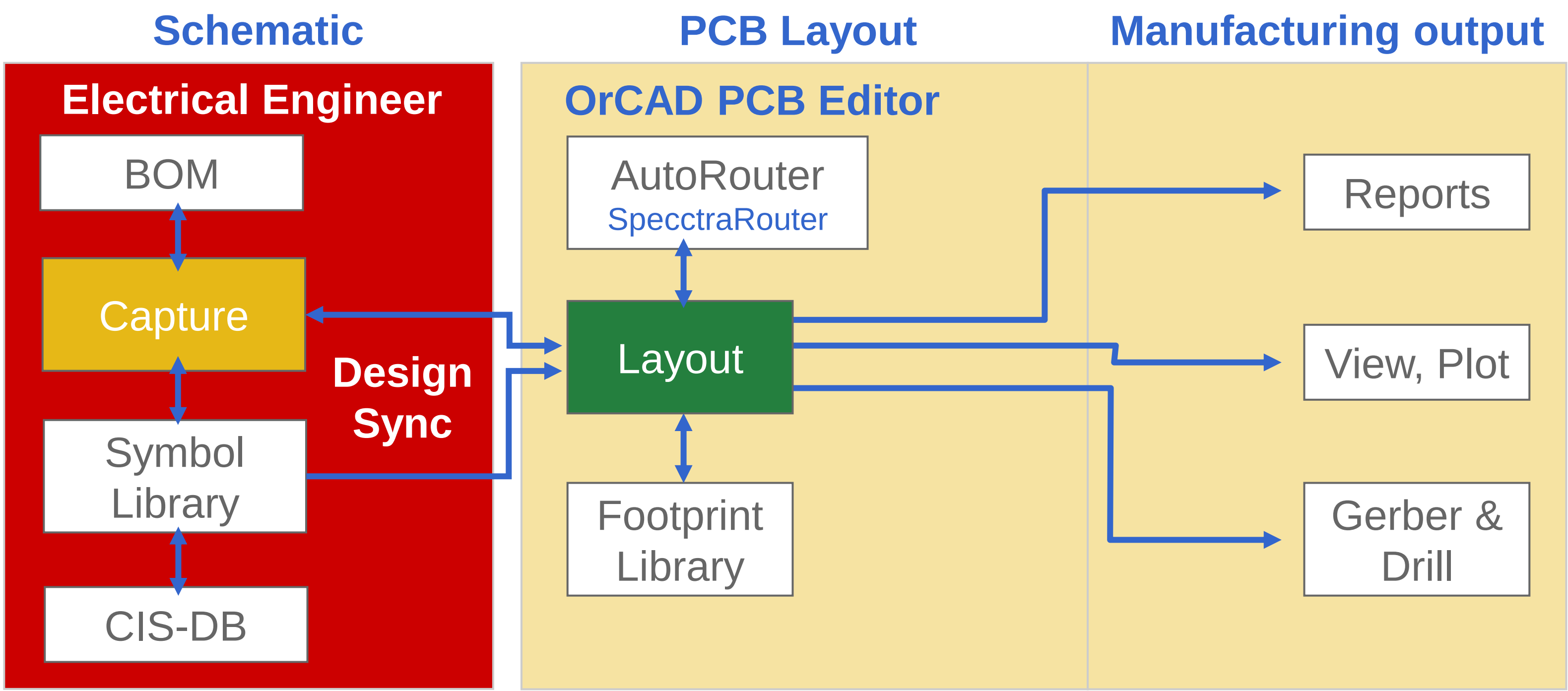
Version 17.4

Table of Contents

Introduction

- This documentation is created for first time users of OrCAD PCB Software. It is neither a training handbook nor a complete user manual.
- Since instructions focus on PCB Editor only, there are no instructions for schematic entry. Start point for this quick start is an already completed OrCAD Capture schematic. A separate OrCAD Capture quick start document is available.
- Because of compactness of this documentation it is not possible to take up all available commands and their options. Here we reference to extensive online help documentation which is part of installation.
- Based on a simple schematic and related PCB layout we will elaborate most important steps of design flow. First time users of PCB Editor are enabled to complete first tasks independently with minimum effort.

OrCAD PCB Design Flow



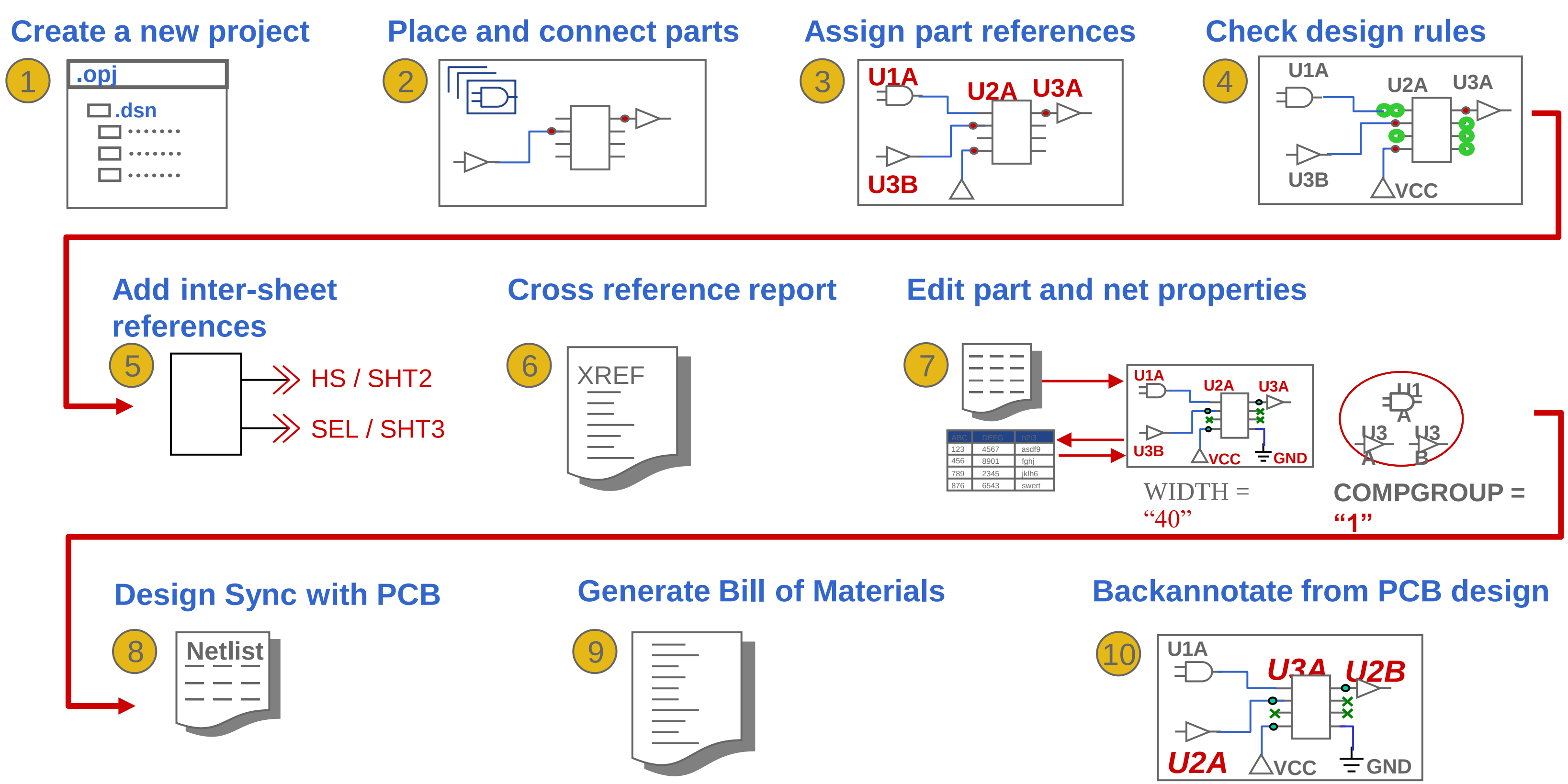
- As you can see, OrCAD PCB Designer Flow consists mainly of two parts.
  - These are the schematic capture module **Capture** and the layout module **OrCAD PCB Editor**.
- Both modules are supplemented by additional sub packages who represent in each combination an ideal tool, enabling the user to complete all tasks with maximized efficiency.

Necessary Steps in Schematic

Necessary Steps in Schematic

- Steps described in this chapter give a brief overview.
- For Capture and Capture CIS separate quick start documents are available.
- Logic data already exists in training data and can be directly imported into a board-file as described in Lab Import of Logic Information on page 72.
- Logic data required for PCB Editor quick start can be found at: ~\PCB_Editor_Demo_17_4\PCB_Editor_Demo\project2

Capture Design Flow



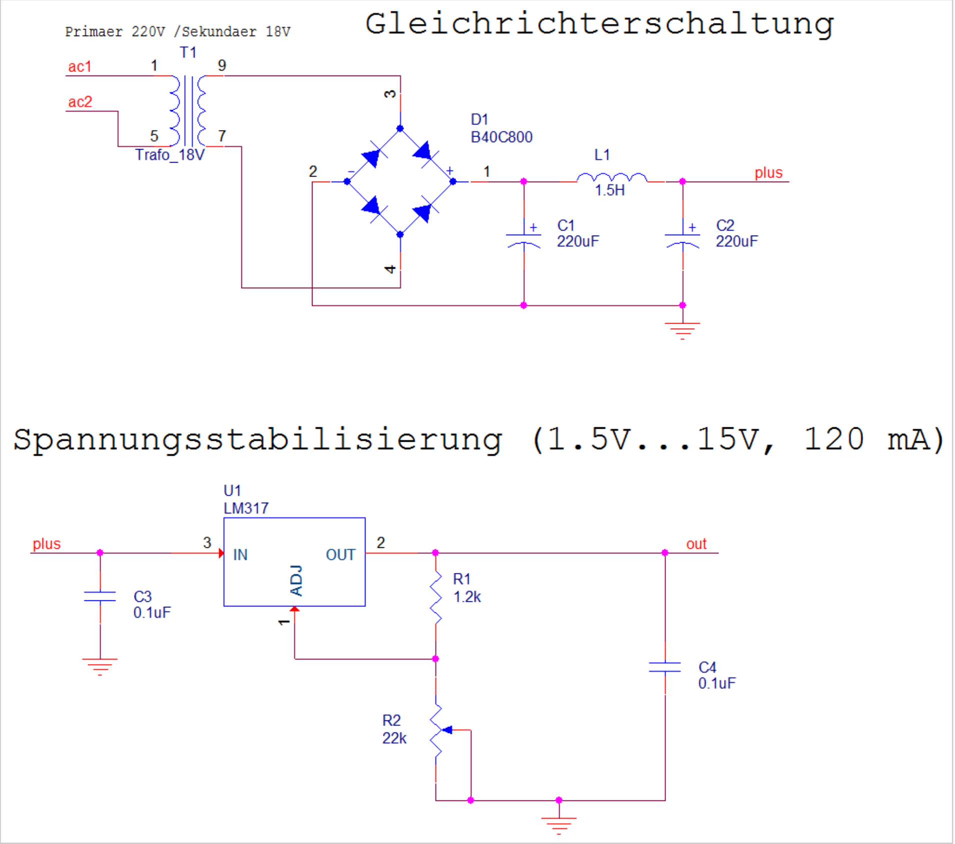
Capture Design Flow

- Graphic on page 7 illustrates typical design flow creating a schematic for following PCB design. Points 1 to 10 illustrate individual work steps in design flow focusing on next pages.
- Main task starts with project creation and is completed with PCB layout synchronization. For sure there are other tasks during design process necessary like report generation, bill of material generation, etc.
- Like wise it can happen that required parts are not in library. In this case a not illustrated part creation step in between is necessary. We will focus on this step later.

Schematic Template

Goal of design flow is, as already mentioned in introduction, to design a PCB layout based on an existing schematic.
Please see below schematic used for our demo example.

More details regarding Capture flow can be found in Quick Start Capture document.



REFDES – Footprints

In list below we have listed footprints of individual parts manually assigned to parts in schematic. Footprints are symbols of electronic parts used in layout tool.

- T1 = ERA-EI30-2_8VA
- D1 = SM_GL_BRUECKE
- U1 = TO220abv
- R1 = SMR_1206
- R2 = VRES34
- L1 = SML_2220
- C1, C2 = Cpol_508
- C3, C4 = SMC_1206

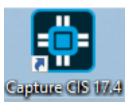
Please ensure correct spelling.

Start of Capture

After starting Capture,
Capture **Session Frame** window will open.

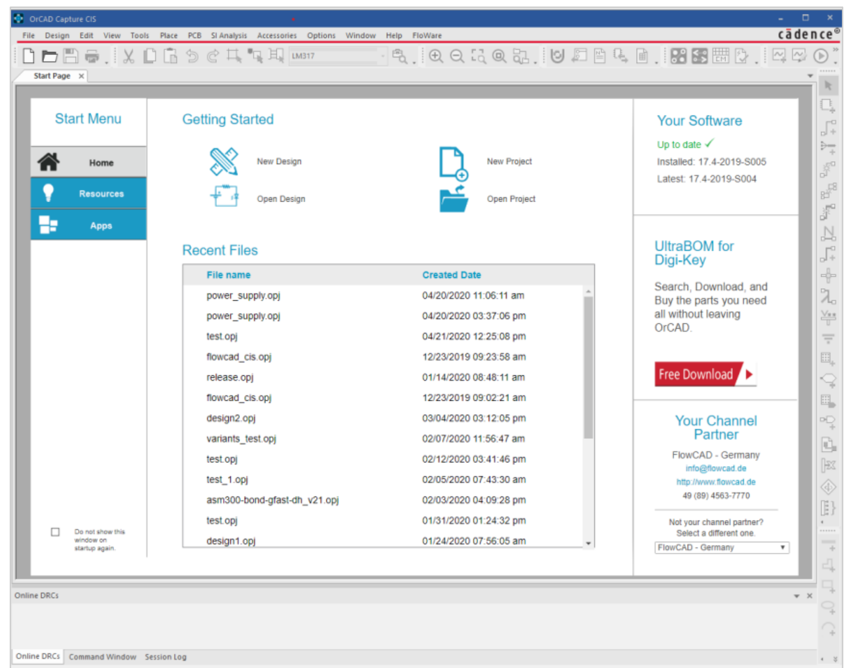
Start via:
Start > All Programs >
Cadence Release 17.4 > OrCAD Products >
Capture CIS

or

Link 

Icon on desktop
At bottom of the window **Session Log** window appears. It can be viewed also in a separate window. All events of current session and messages from other Capture tools are listed here.

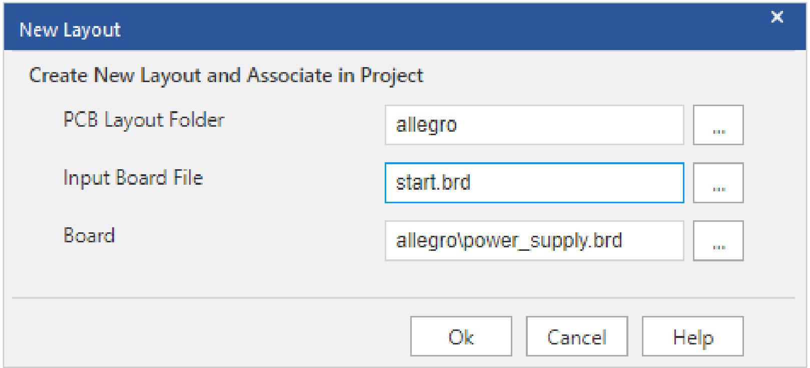
File > Open > Project… will open an existing project, in which design (Power_Supply) is defined.



Layout Creation

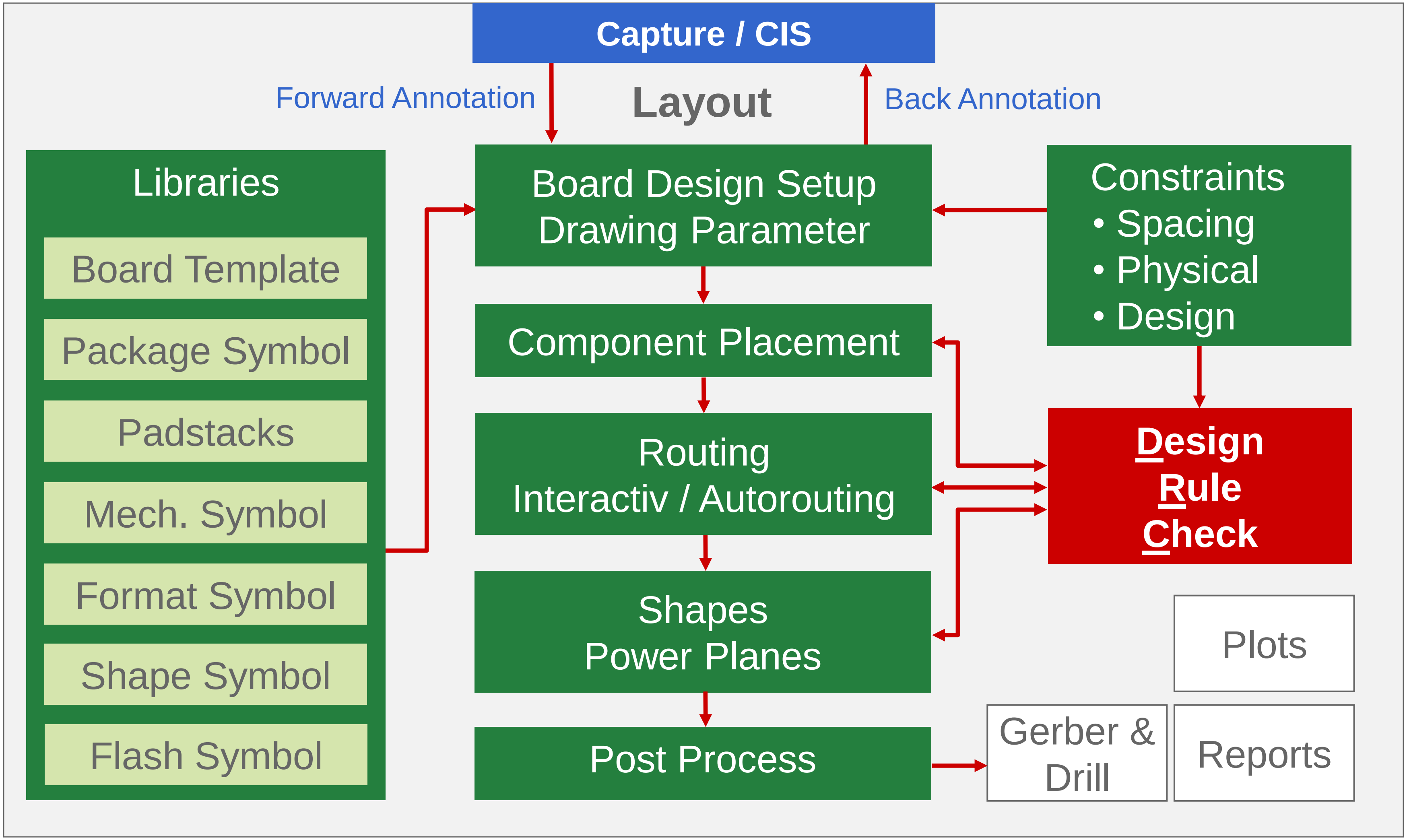
Logic data gets transferred via **PCB > New Layout** into new PCB.

- PCB Layout Folder: Folder for netlist data
- Input Board File: A base or predefined board template
- Board: New generated board file



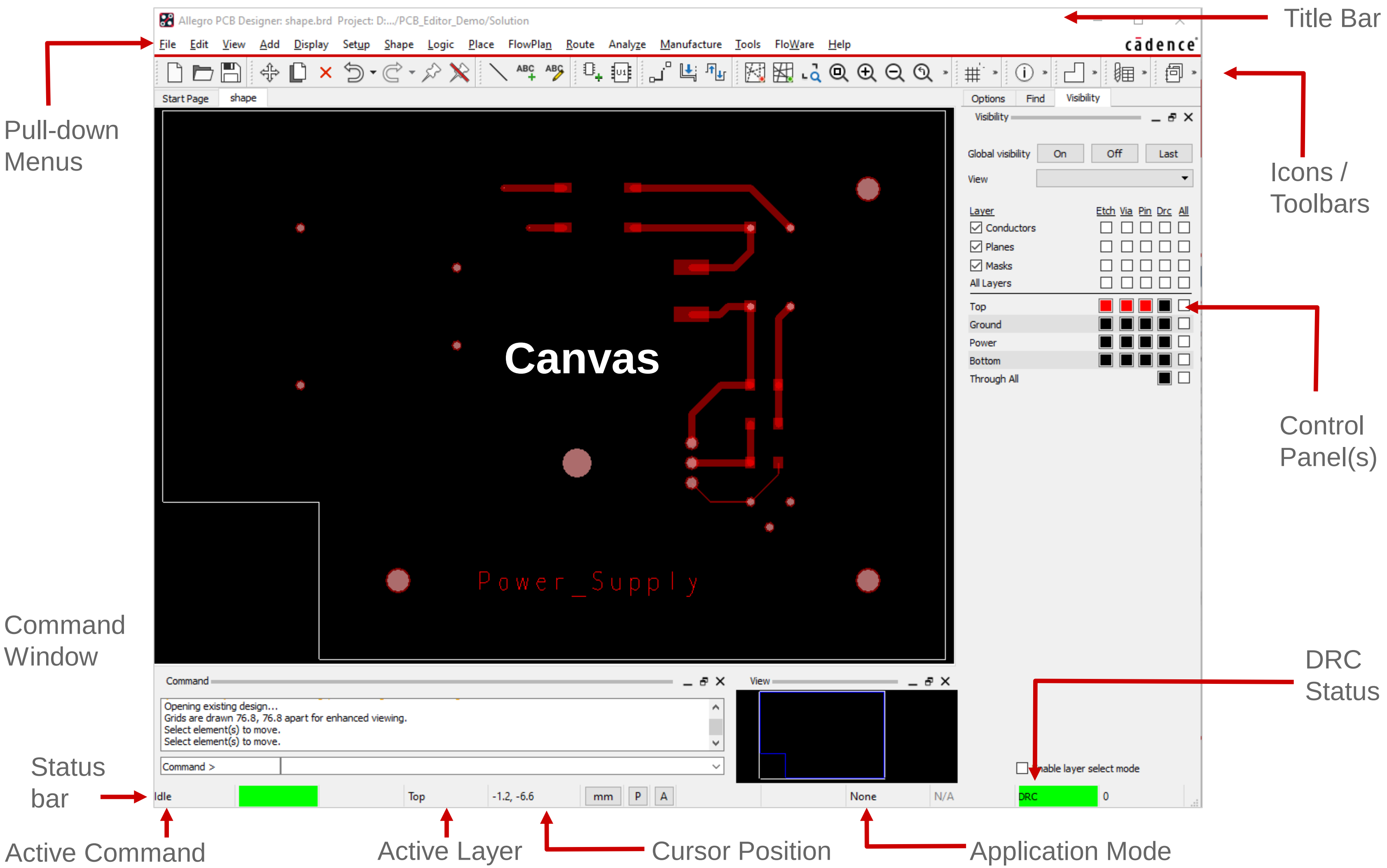
PCB Editor Flow Overview

PCB Editor Flow Overview



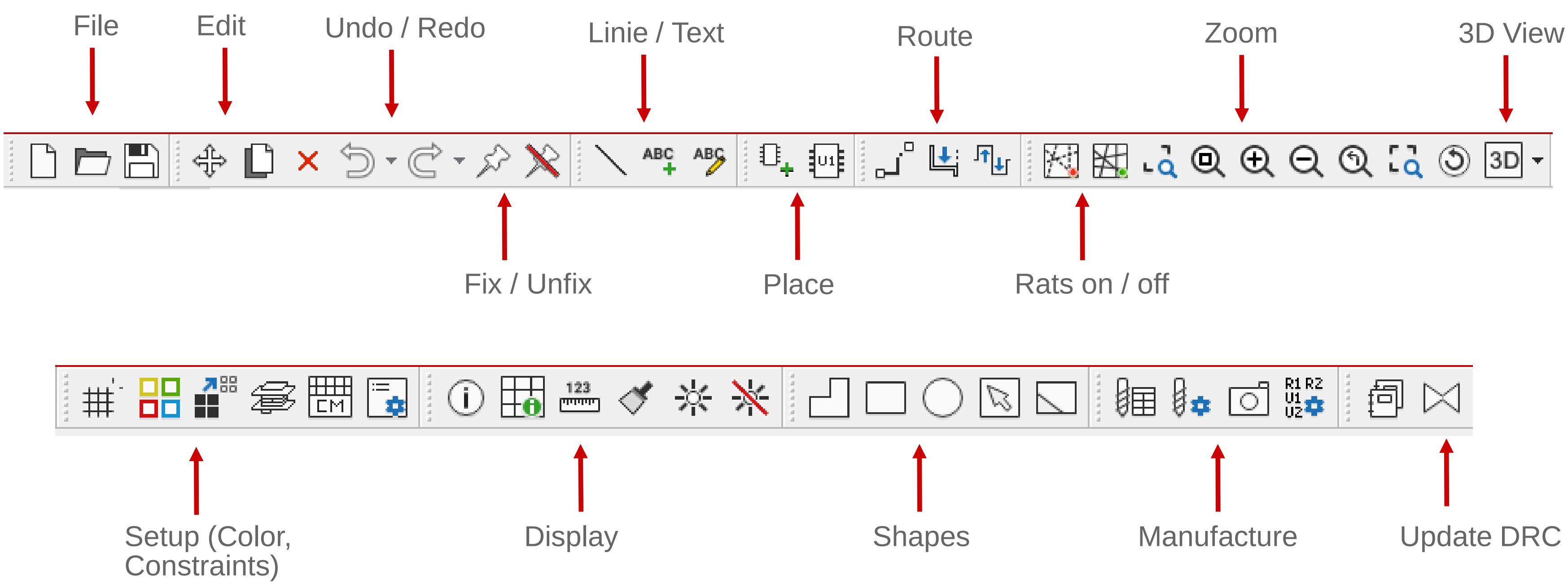
User Interface

PCB Editor and Canvas



Icons and Toolbars

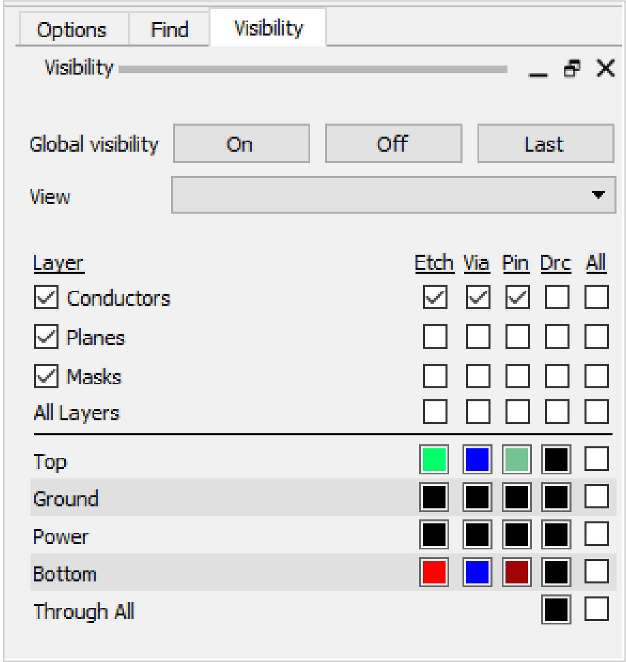
Here you can see all available icons of PCB Editor. They are bundled in groups, so-called toolbars.
View > Customize Toolbar allows to view or hide toolbars.
Like in Windows, toolbars can be arranged along outer edges of window or in a separate location.



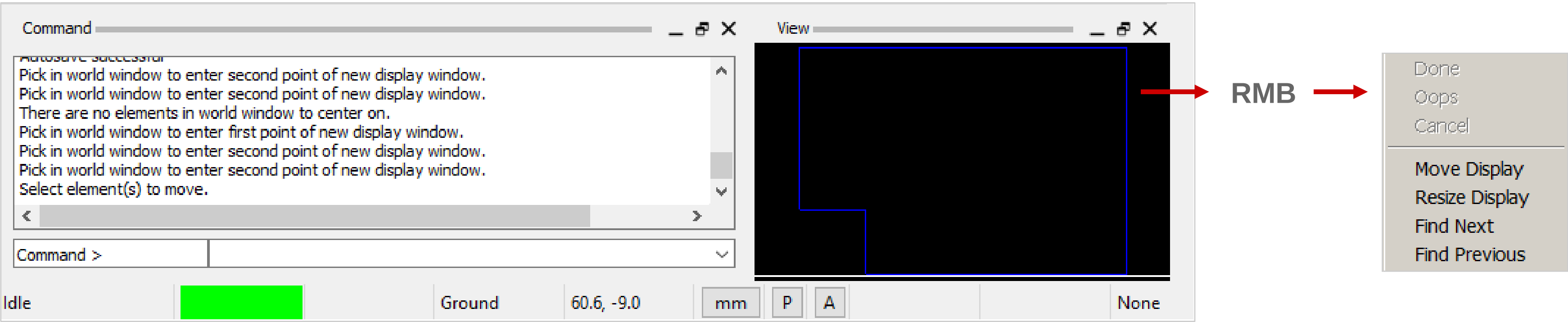
Note
When you hover with curser over an icon, a short description will be displayed.

Control Panels and World View Window (I)

- Options Tab
  - Shows current parameters and values for active command.
  - Shows fields to control active commands.
- Find Tab (Find Filter)
  - Controls which objects can be selected.
  - You can also select objects by entering their name.
- Visibility Tab
  - Controls visibility of routing objects (Etch, Pin, Via, DRC) on conductor and plane layers.



Control Panels and World View Window (II)



View Window

- Shows actual view section relative to entire design.
- Allows active view of partial areas of design.

Command Window

- Allows coordinate / command entry and shows system messages.

Tip

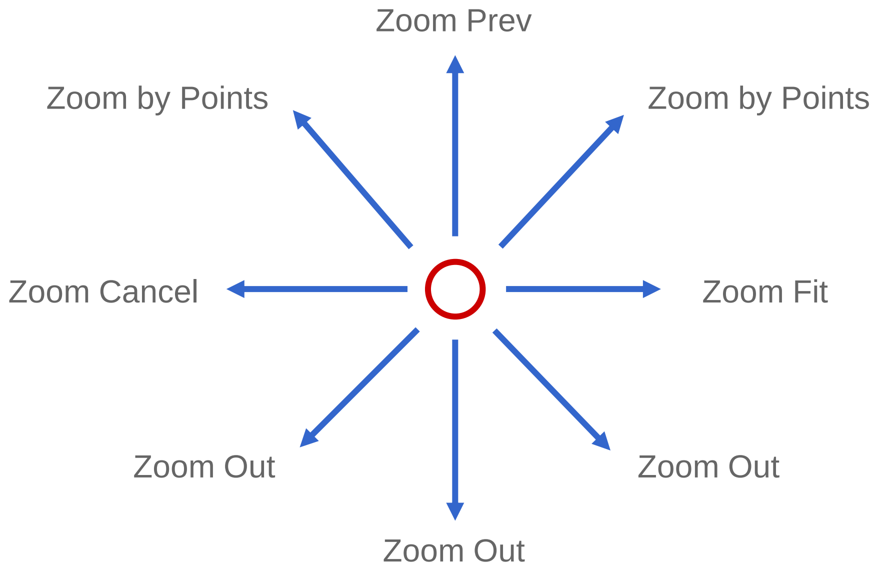
- For larger working area all windows can be closed separately (**View > Windows** or via).

Zoom Control with Middle Mouse Button

Middle mouse button provides you a universal tool to zooming and panning on work window.

1. Circle in the middle represents original selection point. **1st click** with middle mouse button (MMB).

2. Arrows represent direction of mouse movement and following **2nd click** (MMB).



3. Press middle mouse button (MMB) and move mouse. This is the way to pan actual view.

4. A double click (MMB) will execute command **Zoom Center.**

Aliases, Function Keys and Strokes

Aliases, Function keys and strokes allow to execute complete commands or even macros with one click.

Try to perform following examples as alias, function key and stroke:

1. Load board **sample.brd** (folder sample).
2. Type following commands, one after other, in command window and confirm with **ENTER**.
   - **alias Home zoom fit** (Assignment of ZOOM Fit on Pos1-key. Please ensure correct large and lower case).
   - **funckey r iangle 90** (Assignment of 90-degree rotation on R-Key. Please ensure correct large and lower case).
3. Use middle mouse key to zoom into a small area.
4. Press key Pos1 (or Home using an English keyboard). Command Zoom Fit will be executed.
5. Choose **Edit > Move** and select a component.
6. Press key **R** multiple times. Component will rotate by 90 degree every time.
7. Press **RMB > Done.**
8. Press **Ctrl** key and at same time **RMB** – draw a small **Z** over a component.
9. Command **Zoom in** will be executed.

You have successfully defined an alias and a function as well as a stroke function executed.
Type **alias** in command line and ENTER. All default aliases and function keys will be displayed.
Select **Tools > Utilities > Stroke Editor. S**troke editor will start and show all predefined strokes.

Tip
Two assignments above are only present in current session. How to define this kind of assignment permanently, will be explained in next chapter.

Workspace

PCB Editor Data Structure

A Board File (**xyz.brd**) is collection of many drawing layers. Each of these layers can be switched visible or invisible. Each layer can be assigned a color.

PCB Editor is managing these drawing layers within a hierarchy of folders, classes and subclasses. Folders are a collection of classes to support users controlling colors and visibility.

All elements are stored in kind of a 2-level database. First level is referencing to different predefined classes. Some dedicated classes are combined in specific folders.

Folders and classes can neither be deleted nor can new ones be added.

Within each class there are multiple subclasses. Subclasses are second level of database. They are called layers in the design. Predefined subclasses can not get deleted. You can add as many new subclasses as you want. These can get deleted if they do not contain any data.

All routing activities are related to subclasses assigned to class Etch. These subclasses have special DRC properties assigned unlike other classes and subclasses.

For each electrical layer of the board you must add an appropriate subclass. This means, for a 4-layer multilayer you need 4 subclasses under class Etch.

A new defined PCB board is by default generated as a 2-layer board consisting of top and bottom.

Predefined subclasses top and bottom can not get renamed or deleted.

Folders, Classes and Subclasses (I)

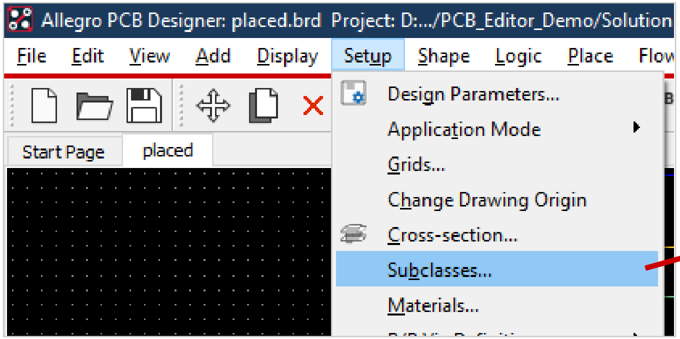
Folder	Classes	Subclasses
Display	Temp Highlight, Grids, Ratsnest (top, bot, thru), Perm Highlight, Waived DRCs, Drill holes, Via Label, Stacked via Label, Background, Pattern, Shading, Transparency	Subclasses nicht vorhanden
Stackup / Conductor	Pin, Via, DRC, Etch, Anti Etch, Boundary	Top, Bottom (and all other user defined PCB board design layers)
Stackup / Non_Conductor	Pin, Via, DRC, Etch, Anti Etch, Boundary	Soldermask_Top, Soldermask_Bottom, Pastemask_Top, Pastemask_Bottom, Filmmasktop, Filmmaskbottom, Through All, Package_Top, Package_Bottom
Areas	Route Keepout, Via Keepout, Package Top, Bottom, Through All, Package Keepout, Package Keepin, Route Keepin, Constraints Region	Top, Bottom, Inner_Plane_Layers, Inner_Signal_Layers, Outer_Layers, Through All
Board Geometry	Board Geometry	Outline, Plating_Bar, Assembly Notes, Tooling_Corners, Dimension, Place_Grid_Top, Place_Grid_Bottom, Top_Room, Bottom_Room, Both_Rooms, Switch_Area_Top, Switch_Area_Bottom, Silkscreen_Top, Silkscreen_Bottom, Assembly_Detail, Soldermask_Top, Soldermask_Bottom, Off_Grid_Area, NcroutePath, Wb_Guide_Line

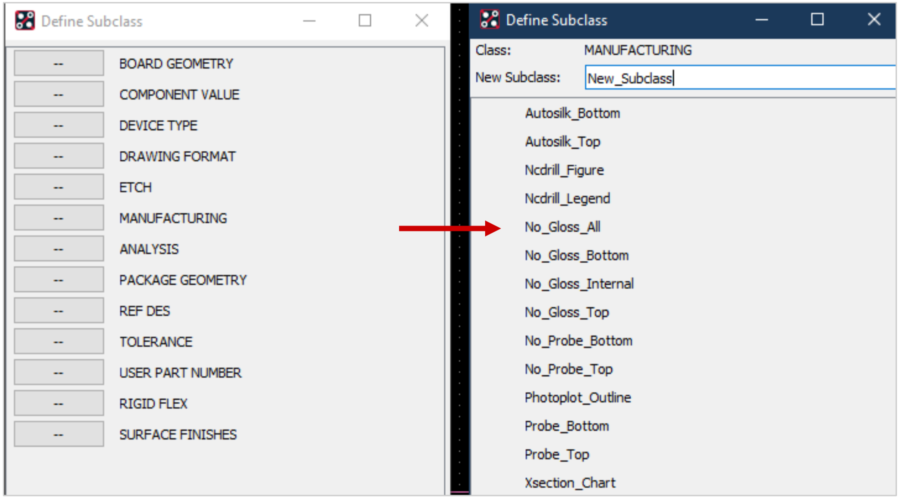
Folders, Classes and Subclasses (II)

Folder	Classes	Subclasses
Package Geometry	Package Geometry	Assembly_Top, Assembly_Bottom, Place_Bound_Top, Place_Bound_Bottom, Pin_Number, Pad_Stack_Name, Silkscreen_Top, Silkscreen_Bottom, Body_Center, Soldermask_Top, Soldermask_Bottom, Display_Top, Display_Bottom, Modules, Dfa_Bound_Top, Dfa_Bound_Bottom, PasteMask_Top, PasteMask_Bottom
Embedded Geometry	Embedded Geometry	All
Components	Comp Value, Device Type, Ref Des, Tolerance, User Part Number	Assembly_Top, Assembly_Bottom, Display_Top, Display_Bottom, Silkscreen_Top, Silkscreen_Bottom
Manufacturing	Manufacturing	Autosilk_Top, Autosilk_Bottom, Ncdrill_Legend, Ncdrill_Figure, No_Gloss_All, No_Gloss_Top, No_Gloss_Bottom, No_Gloss_Internal, No_Probe_Top, No_Probe_Bottom, Photoplot_Outline, Probe_Top, Probe_Bottom, Xsection_Chart
Drawing Format	Drawing Format	Drawing_Origin, Outline, Revision_Block, Revision_Data, Title_Block, Title_Data
Analysis	Analysis	Low_Isocontour, Medium1_Isocontour, Medium2_Isocontour, Medium3_Isocontour, High_Isocontour, Pcb_Temperature

Folders, Classes and Subclasses (III)

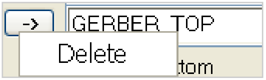
To add additional layers (subclasses) to PCB, desired class should be selected in pull-down menu **Setup > Subclasses. N**ame of new subclass can be defined in subsequent windows.





With exception of routing layers (Etch) all user defined layers can be entered via these menus. When you choose **ETCH**, Layer Stackup will be started automatically to define additional routing layers. More details on this topic can be found at page 146 under [Masterboard].

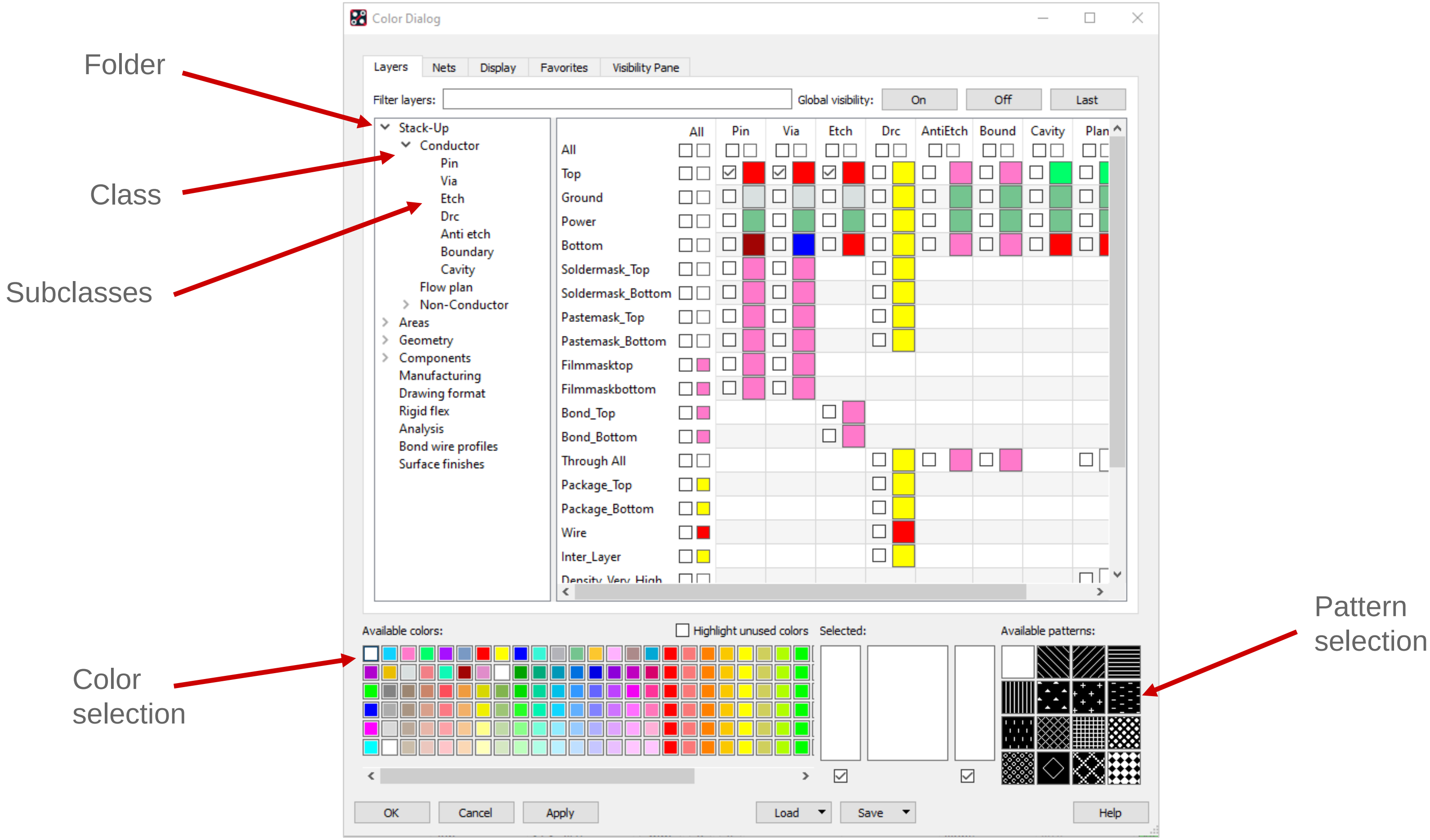
All user defined layers have a white background to highlight user defined layers in menus. All other layers are default layers of system. One click on arrow button will delete a user defined layer.



Only layers without any data can be deleted.

Control of Color and Visibility (I)

Via **Display > Color > Visibility** or visibility and color of individual layers (classes / subclasses) can be set.



Control of Color and Visibility (II)

Via **Display > Color > Visibility** or visibility and color of individual layers (classes / subclasses) can be set.

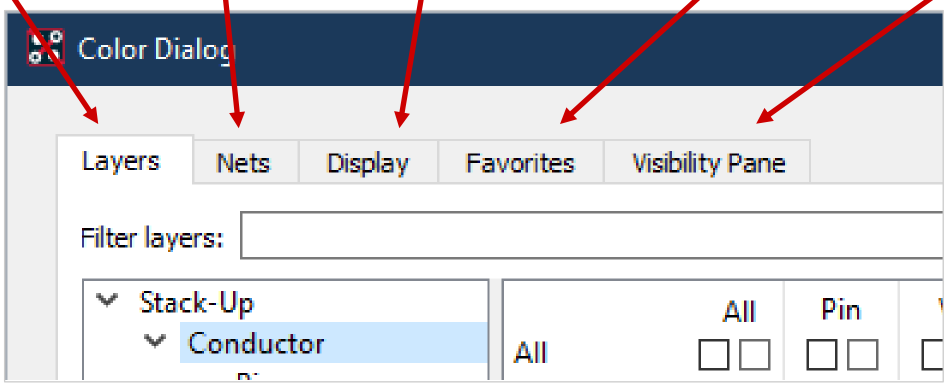
Color / Visibility of design layer

Net colors

General display settings of working area

User defined favorites (via RMB click in layer tab)

Definition of Visibility Pane



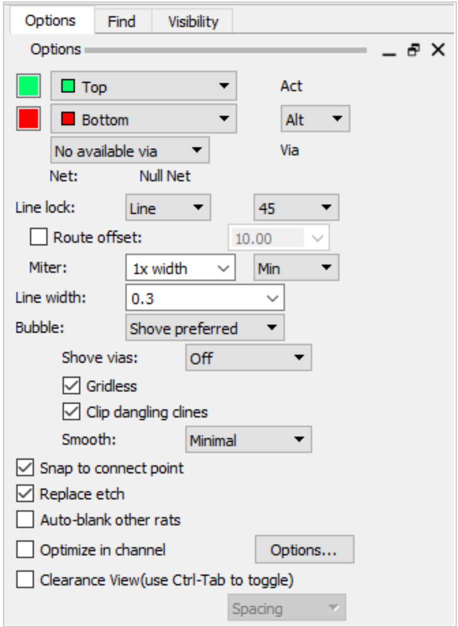
Tip

In display settings you also have functions like shadow mode and transparency for individual appearance of design.

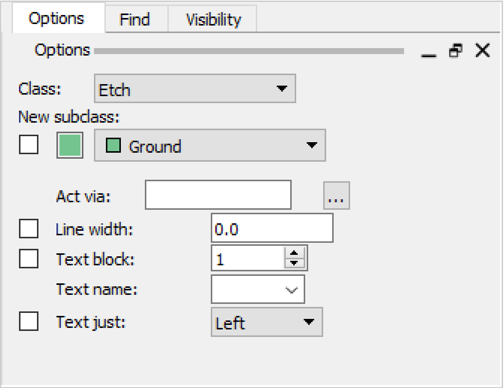
Option Window of Control Panel

Parameters in option window depend on respective command and offer a variety of options for respective commands. You should therefore always keep an eye on window option during interactive work.

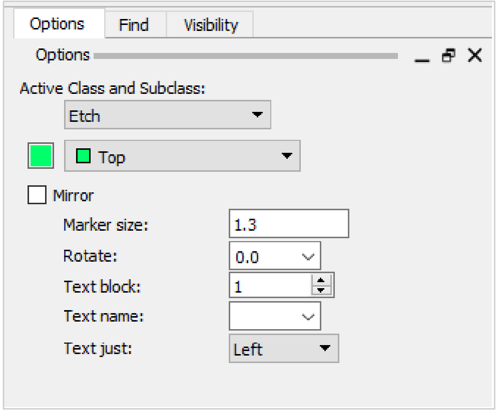
Route > Connect



Edit > Change



Add > Text

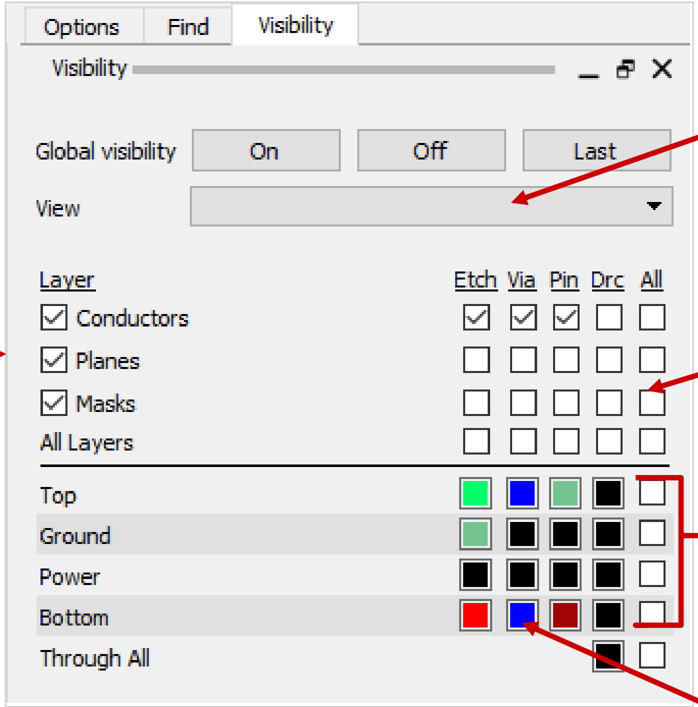


Changes take effect immediately. They also overwrite pre-settings made in PCB Editor setup. For example via **Setup > Design Parameters… > Design > Linelock / Symbol.**

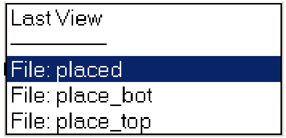
Visibility Control Pane

Visibility control panel provides a fast method to switch single layers on or off. Layer control related to individual elements applies only to **copper and mask layers.**
For documentation layers, color dialog window or user defined **color views** can be used.

Exclude / Include Layers



User defined color views



Control conductor, plane and mask layers

Individual layer control

Individual control of elements

User Preferences

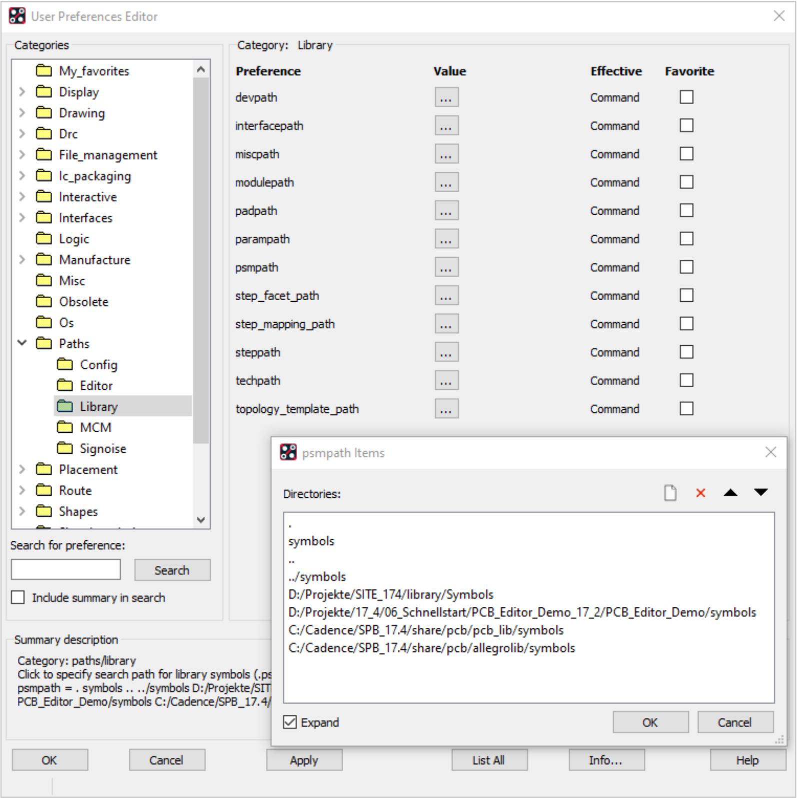
Via **Setup > User Preferences…** behavior and settings of PCB Editor can be predefined.

One of most important settings is **psmpath** shown in the picture. This and **padpath** decide whether library elements are found during placement or not.

Picture shows default settings. These should not be modified at beginning.

All PCB Editor default settings are stored to a file named **env** located under **< your_Installation > /share/pcb/text.**

As soon as you open illustrated menu and perform a modification, a folder named **PCBENV** and a file **env** inside will be created in your home directory.
In this file you can define your individual aliases and function keys. This file will be automatically loaded with every new start of PCB editor.



Library

Library Elements

Libraries that are required for layout can contain different elements. A short overview and explanation regarding these elements is shown below.

- **Package symbol (.psm)** footprint (DIP14, SOIC16, etc.), package symbol
- **Padstack (.pad)** Pad and drill definition of pins on each layer
- Mechanical symbol (.bsm) Mechanical symbol, i. e. spacer, predefined outline of a board with mounting holes
- Format symbol (.osm) Drawing frame for documentation, can not have pins
- Shape symbol (.ssm) Predefined Copper shape, i. e. for special pad shapes
- Flash symbol (.fsm) Copper shape for thermal ties on plane layers
- Board template (.brd) Board template with outline, technology (stackup, spacings, …) and more pre settings

Padstack and package symbol are explained on following pages.

All listed symbols modified with **symbol editors** are edited and saved as .dra.
After compilation (.psm, .bsm, .osm, .ssm, .fsm) available for layout consumption.

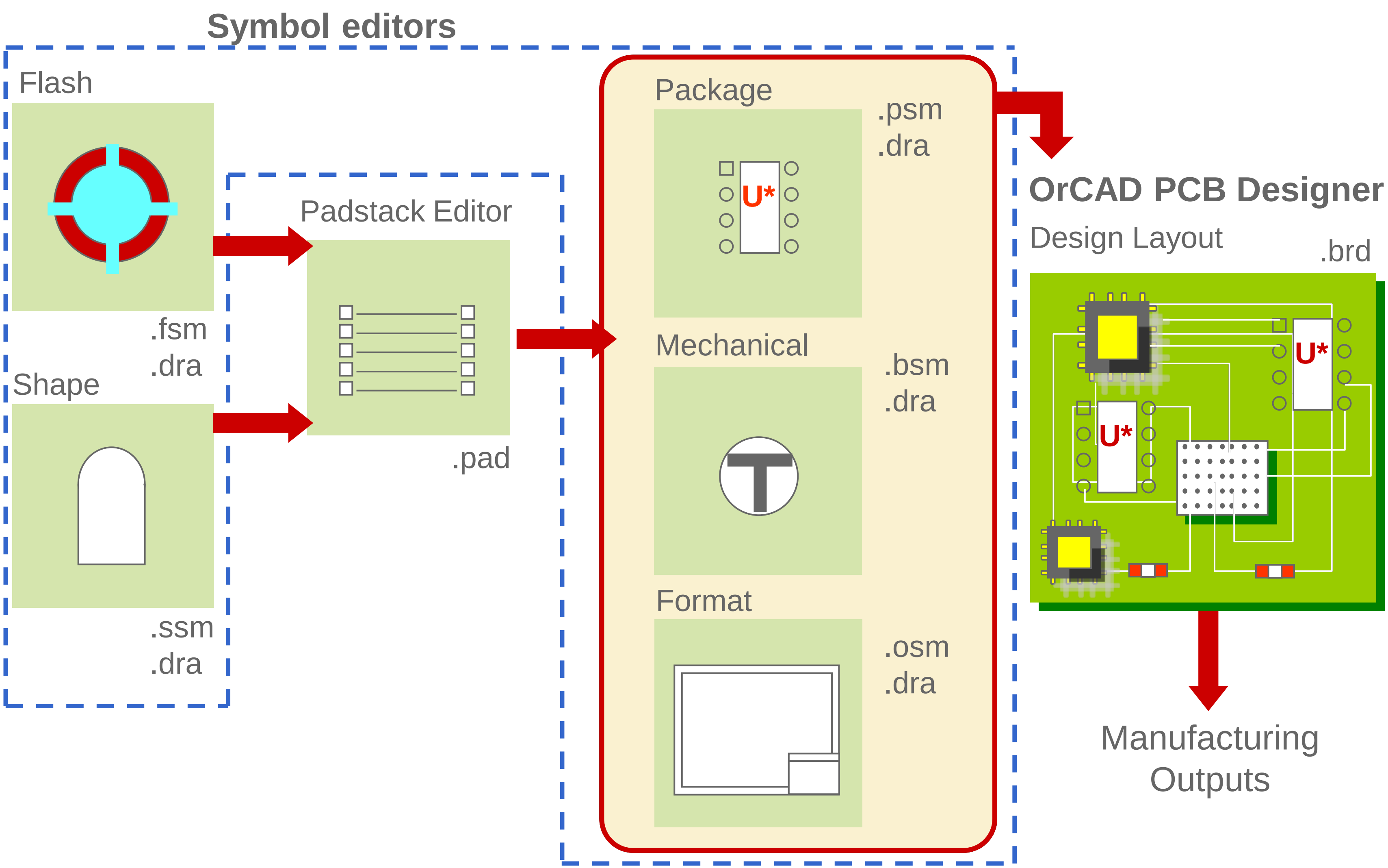
All symbol editors are based on PCB Editor and have same use model. Only function set is different.

Since symbol type is part of .dra file, correct symbol editor will open for editing.

There is a separate **Padstack Editor** for padstacks.

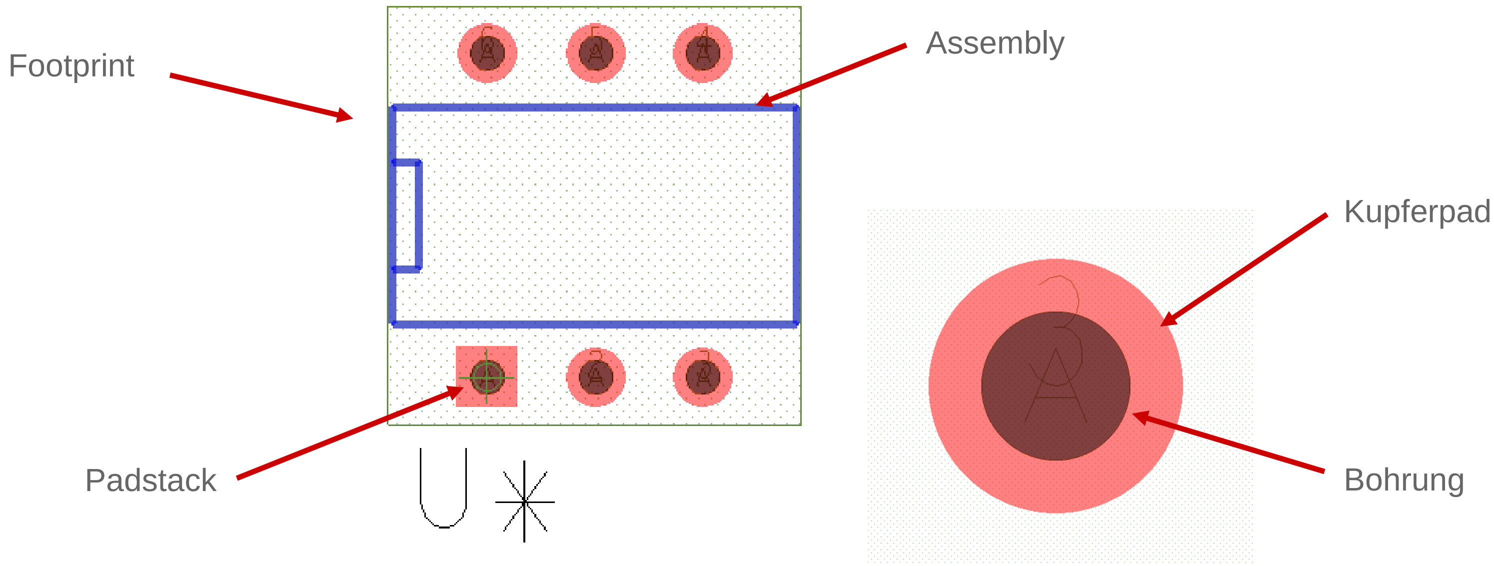
You will find an overview of different symbol editors on next page.

Editor Overview



Footprint as a Basic Symbol

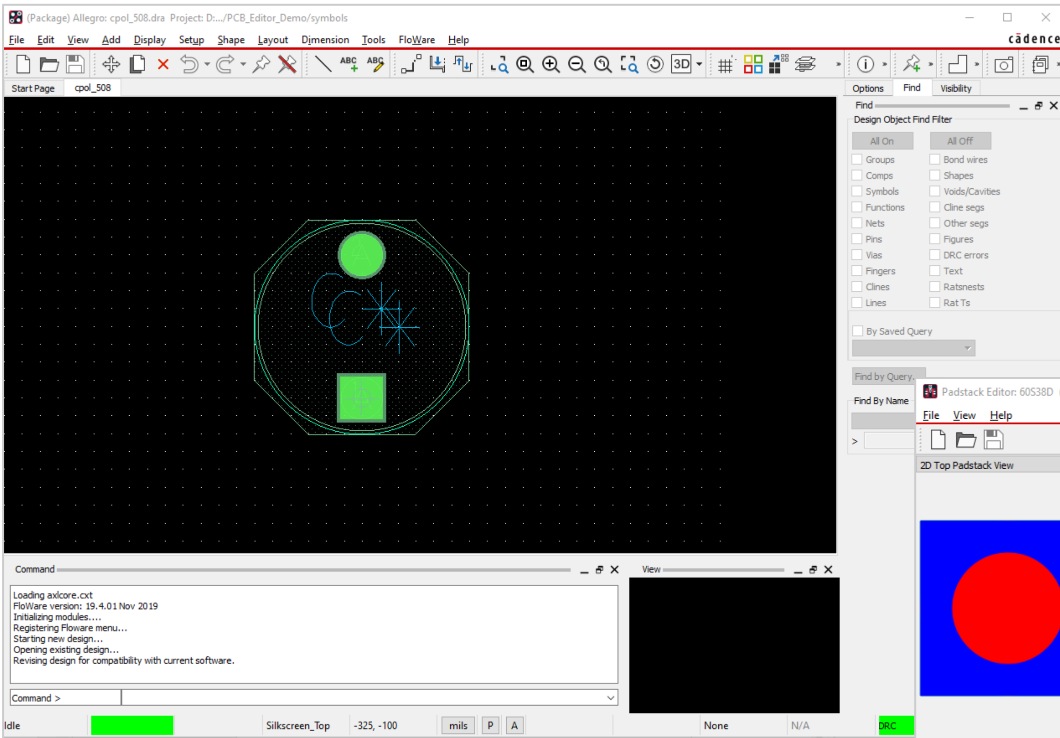
Symbol editors are used for necessary layout library elements.
Most important element – the **footprint** – will be representative example.



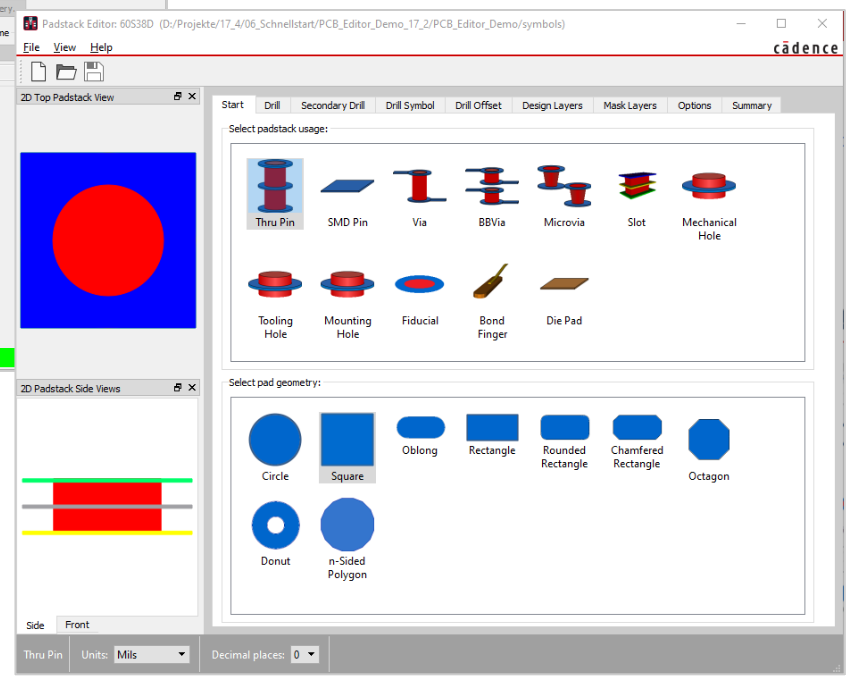
There are also pure graphic symbols for documentation or production purposes (company logos, drawing frames, adjustment markers), which can also be created using one of the symbol editors.

Library Tools

Symbol Editor



Pad Editor



Padstacks

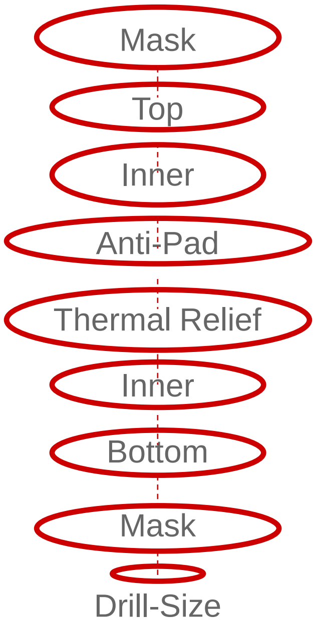
Padstack (Structure)

Most important element of a footprint is padstack.

A padstack contains geometric shape of a pad on every layer of a board as well as drill diameter for thru hole components.

There are basically two types of padstacks:

Through-Hole Padstack



Surface-mount Padstack

Definition of pad size for all layers (electrical or nonelectrical) is done in padstack editor. Electrical layers are all signal and plane layers.

Nonelectric layers are solder mask, paste mask and film mask. Film mask can be used for multiple purpose.

Default routing layers are start layer, default internal and stop layer.

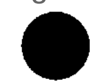
The **DEFAULT INTERNAL** definition is used for additional inner layers besides top and bottom.

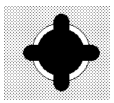
Anti-pad and thermal-relief are only found on negative plane layers (negative copper layers on inner layers; please see stackup definition).

Padstack (Details)

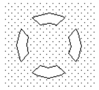
- **Regular Pad**
  Standard pad with a regular pad shape (circle, square, rectangle, oblong, octagon). Only on positive layers.
- **Thermal Relief, Positive**
  Used to connect pins on positive layers to copper areas. This pad is a combination of regular pad and physical rules / clearance rules.
- **Thermal Relief, Negative**
  Will use a flash to connect a pin on a negative plane.
- **Anti-Pad**
  An anti-pad isolates a pin from surrounding negative copper area.
- **Shape**
  Non regular pad described by a polygon (shape) in shape editor.

regular

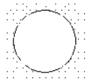




TR_80_60



anti-pad



Tip
Please pay attention to always define all pad types (regular, thermal and anti-pad) for all routing layers. With this method you define a general-purpose padstack.

Pad Designer – Start

There are two ways to start Pad Designer :

- **Start > Cadence PCB Utilities > PCB Editor Utilities 17.4-2019 > Padstack Editor 17.4**

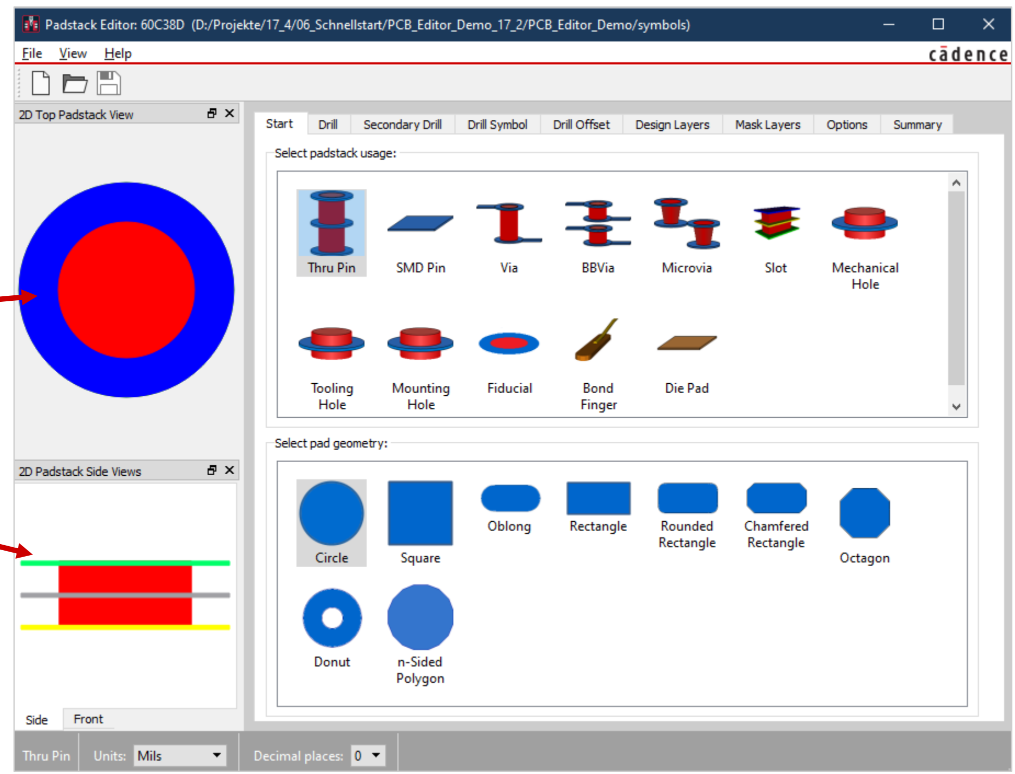
or directly from PCB Designer resp. symbol editor:

- **Tools > Padstack > Modify Design [Library] Padstack…**

On **start** page you can choose different padstack types.
Based on selection next tabs are available and prepared.

Preview top

Preview side view

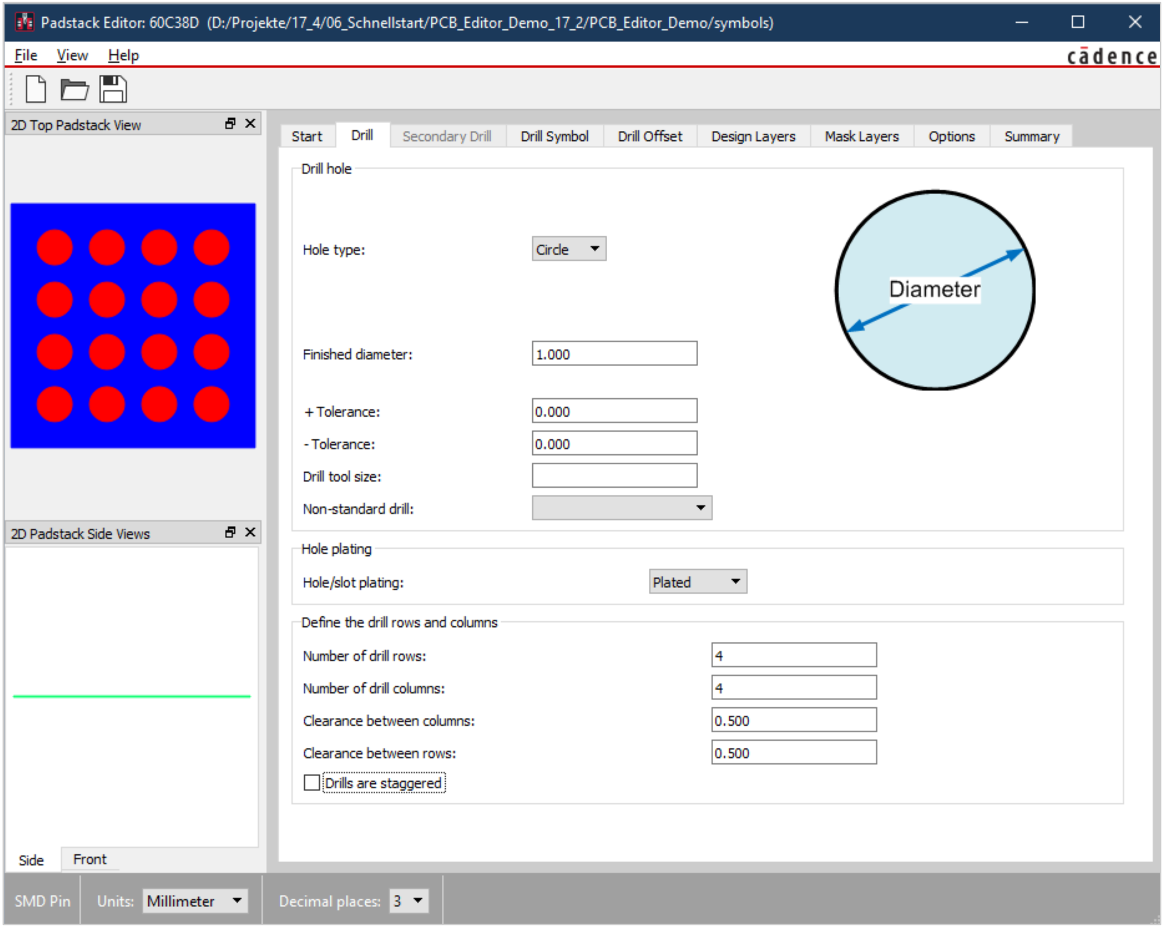


Pad Designer – Drill

In tab **Drill** you have access to all necessary settings to define drill holes.

In addition to drill diameter and tolerances, drill type (e.g. laser) can also be specified.

Furthermore, it is possible to define drill matrices.

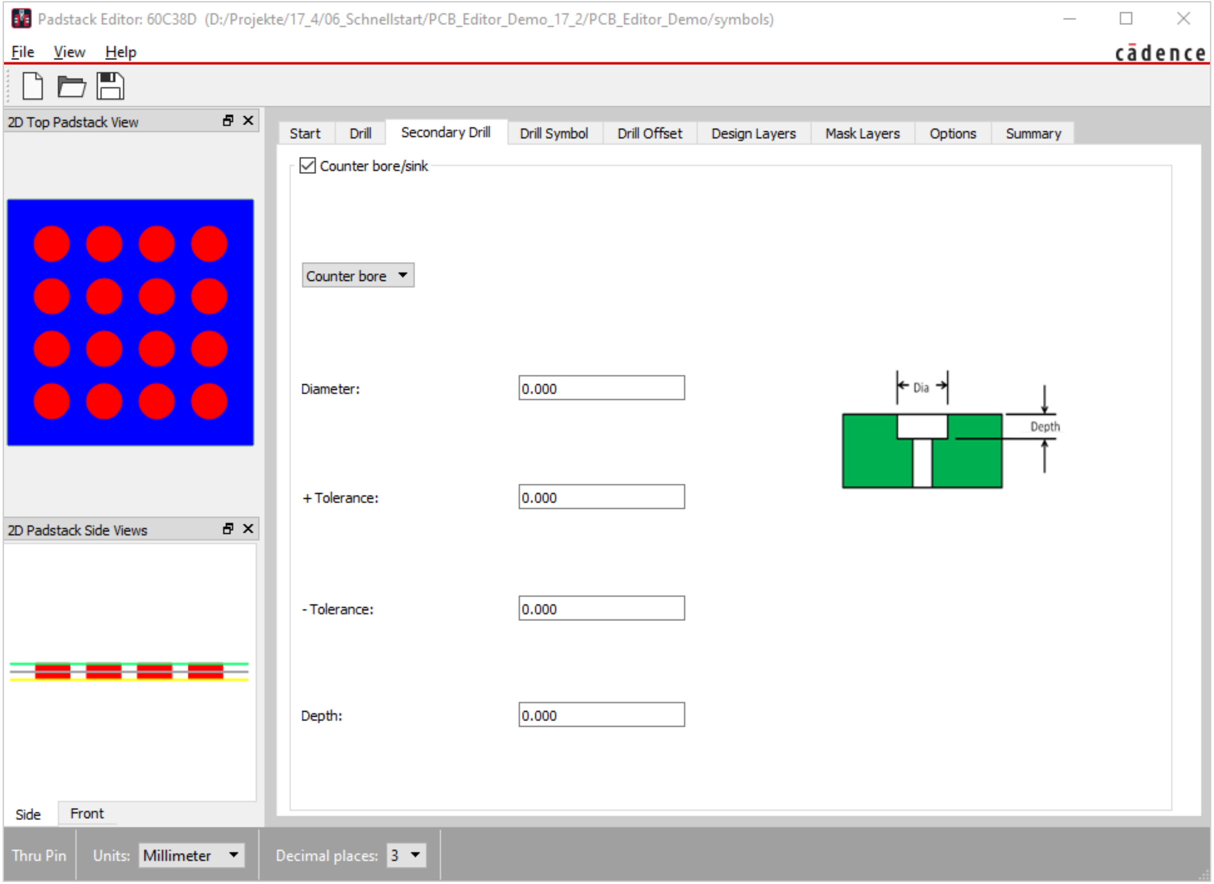


Pad Designer – Secondary Drill

In tab **Secondary Drill** you can define all necessary settings for Secondary Drill and Back drilling.

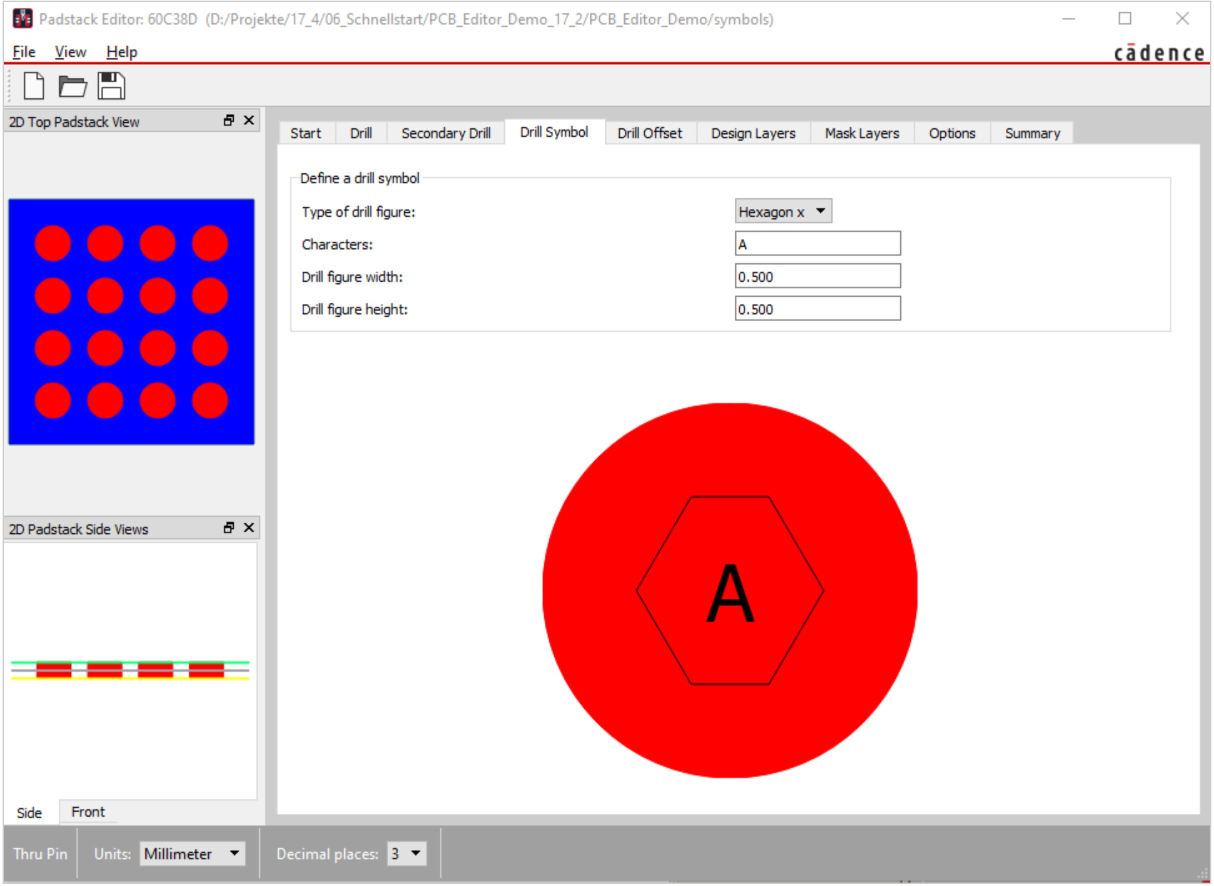
In addition to diameter and tolerances it is possible to define an individual symbol for back drills.

Note
Back drilling is supported in Allegro PCB only.
OrCAD PCB Editor allows secondary drills.



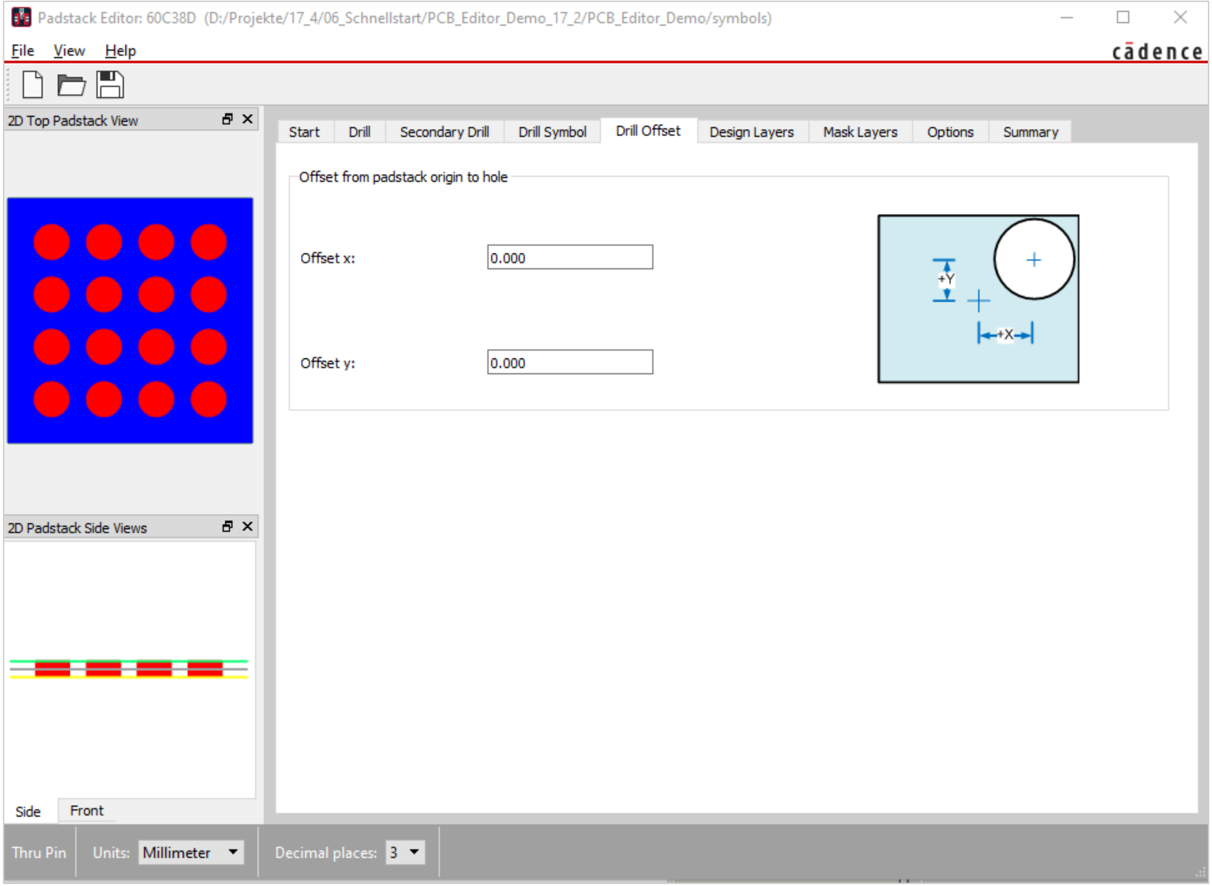
Pad Designer – Drill Symbol

In tab **Drill Symbol** you can define drill symbols for drill table on manufacturing documentation.



Pad Designer – Drill Offset

In tab **Drill Offset** you can define an offset between pad and drill hole.

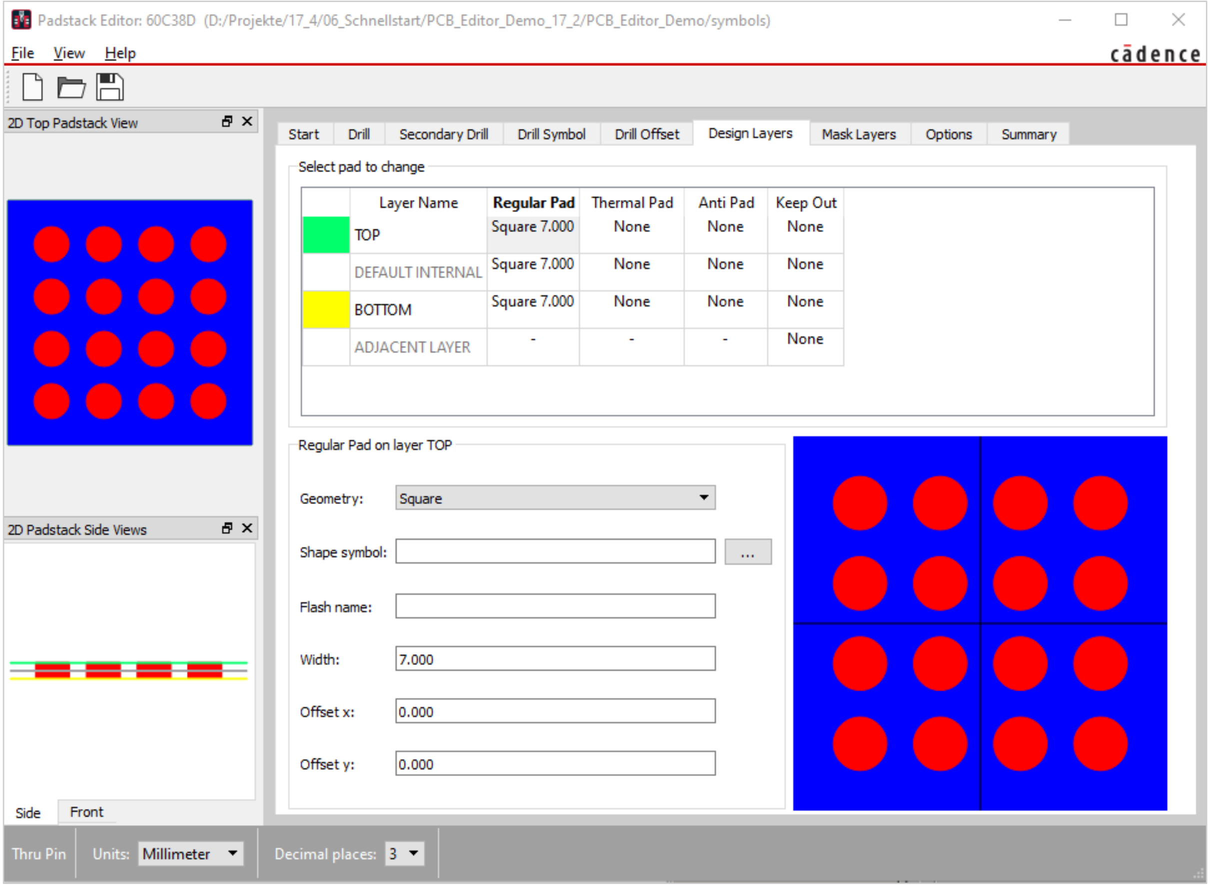


Pad Designer – Design Layers

In addition to tabs for defining holes, Design Layers tab is most important in Padstack Editor.

Here you define pads, copper areas, and associated structures like anti-pads.

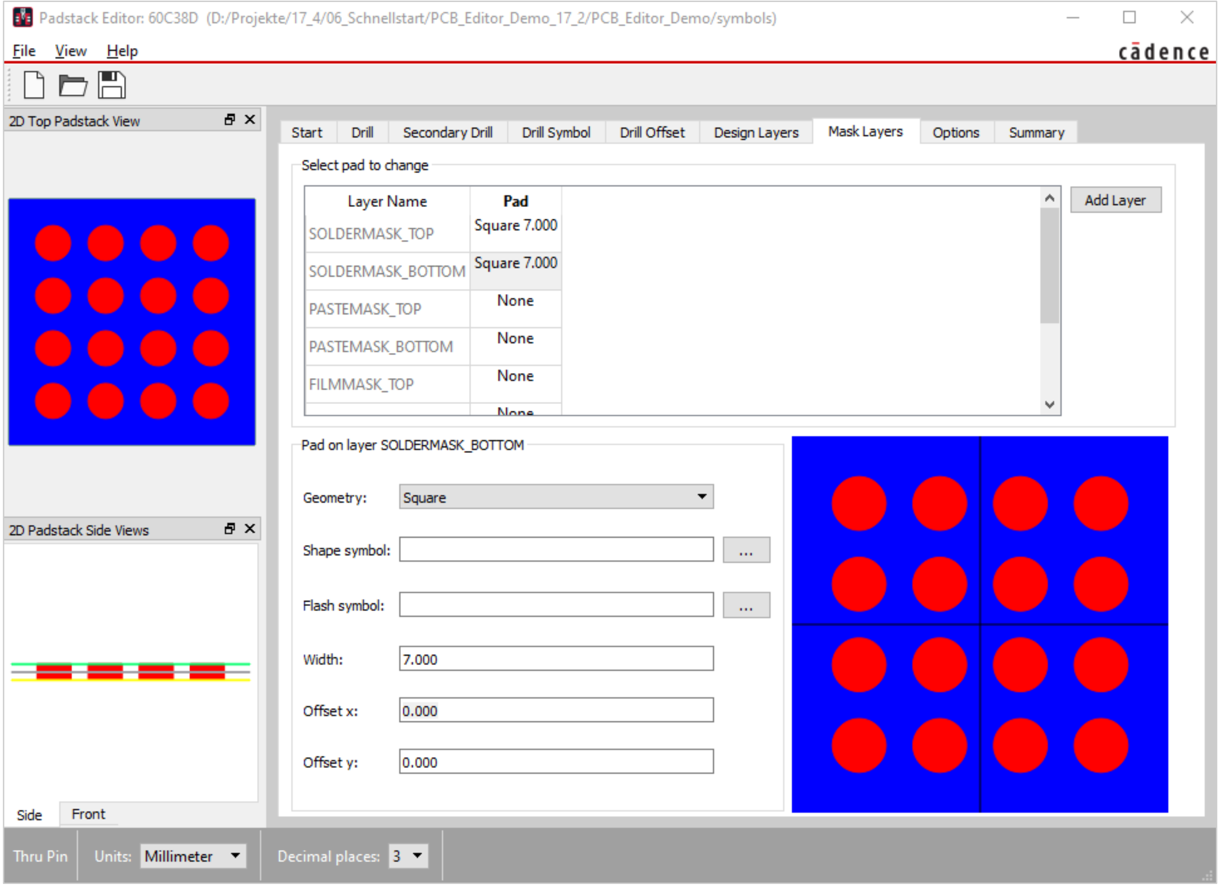
Similar to preview on left side, a preview of selected structure is available in lower right corner.



Pad Designer – Mask Layers

In tab **Mask Layers** you can define different mask layers.

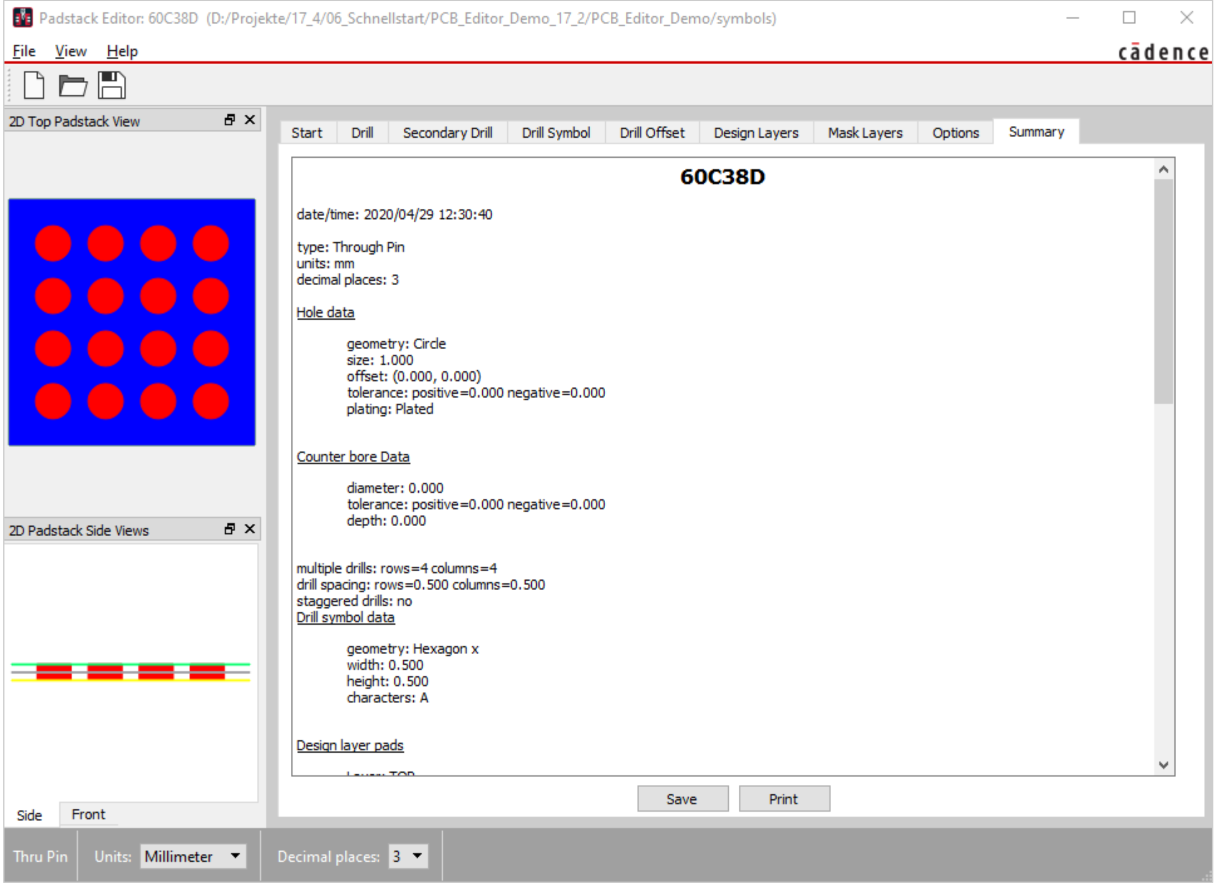
In addition to gap in solder resist, you can define solder paste and if necessary, gaps for other mask layers.



Pad Designer – Options / Summary

In **Options** tab more additional options are bundled.

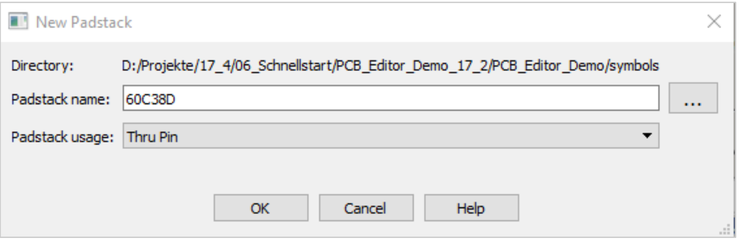
Summary tab shows a summary of different layers, drills and additional settings. For documentation purpose this summary can be also exported in html.

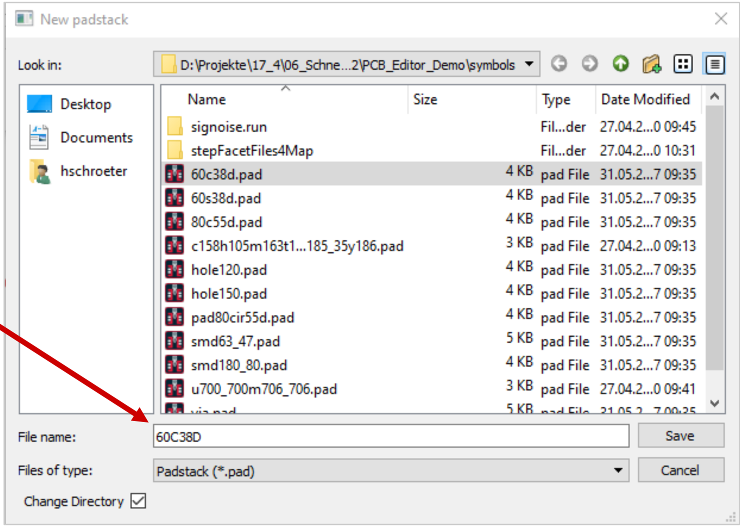


Lab: Padstack (Start)

Next, we will show sequence of steps to generate a padstack. We will create a through hole padstack.

1. **Start > Cadence PCB Utilities > PCB Editor Utilities 17.4-2019 > Padstack Editor 17.4**
2. **File > New** in Padstack Editor.
3. Browser window on the right will appear.
4. Navigate with browser button to your target folder (in this case **Play**).
5. Enter padstack name **60c38d**.
6. Activate change directory box. The chosen directory will be your working folder.
7. Please **save** new padstack.
   By selecting check box under 5. play will become the current working folder.
8. Headline of pad designer shows name and path.





Note
Naming convention of Cadence is referenced to units in mil:
60c = 60 mil circle; **38d** = 38 mil hole (final diameter); in addition: **r** = rectangle; **s** = square; **o** = oblong
This notation is used for all padstacks delivered by Cadence and is easy to recognize.

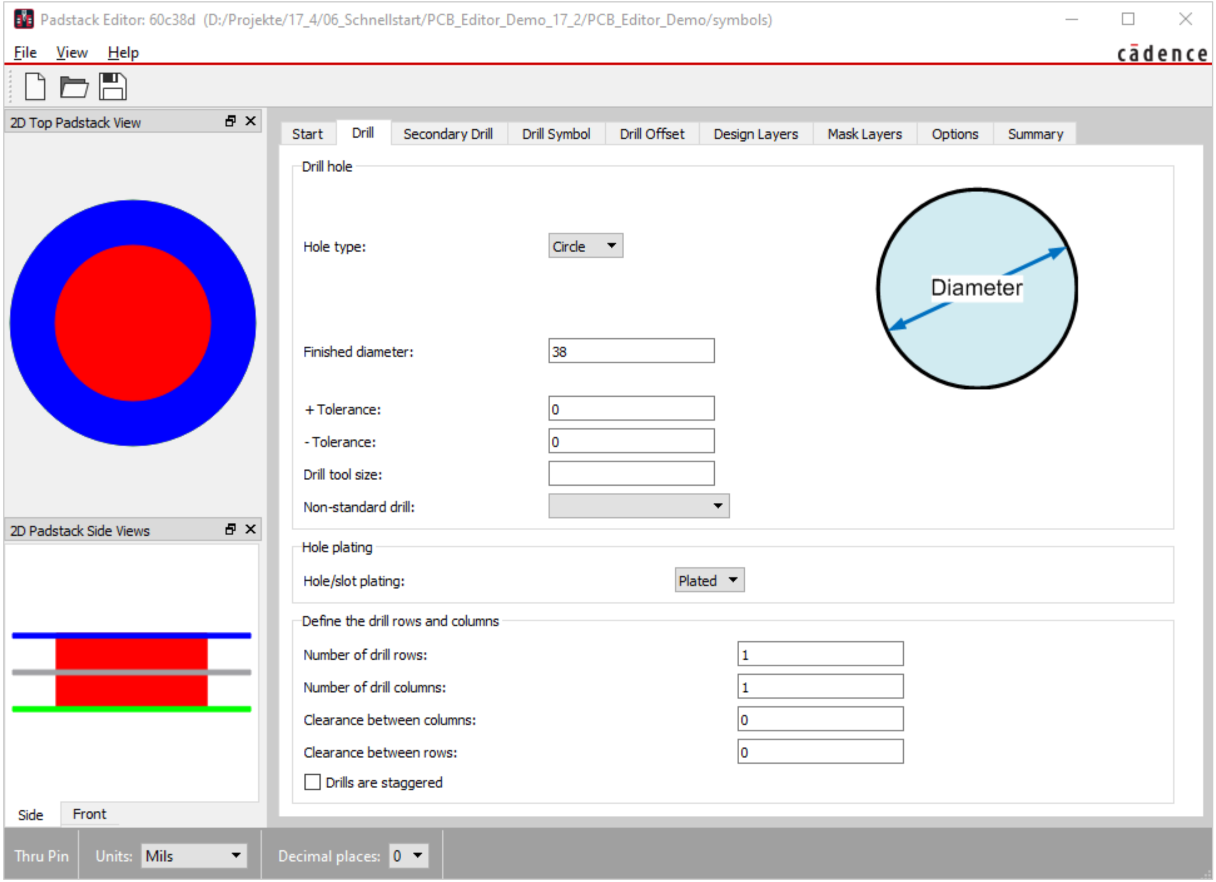
Lab: Padstack (Drill)

Please enter values from the picture into **Drill** tab.

Tip

To ensure to enter all necessary entries, please move from left to right through all tabs.

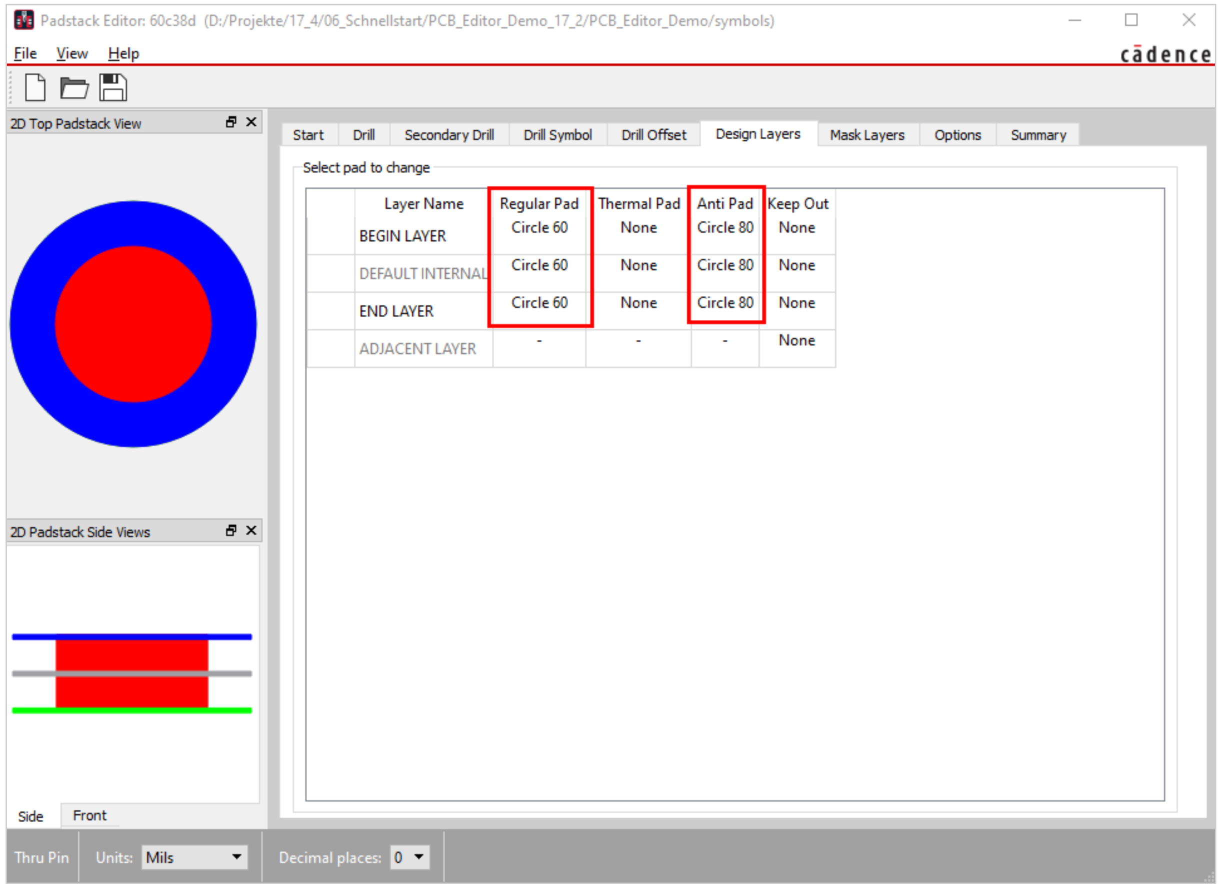
All unused tabs of a padstack type are grayed out.



Lab: Padstack (Design Layers)

Please enter values from the form into **Design Layers** tab.

Create pads in **Mask Layers** tab.



Tip

Via copy / paste, pads can be copied from one to another layer within a tab.

Symbols

Overview Footprint Design Process

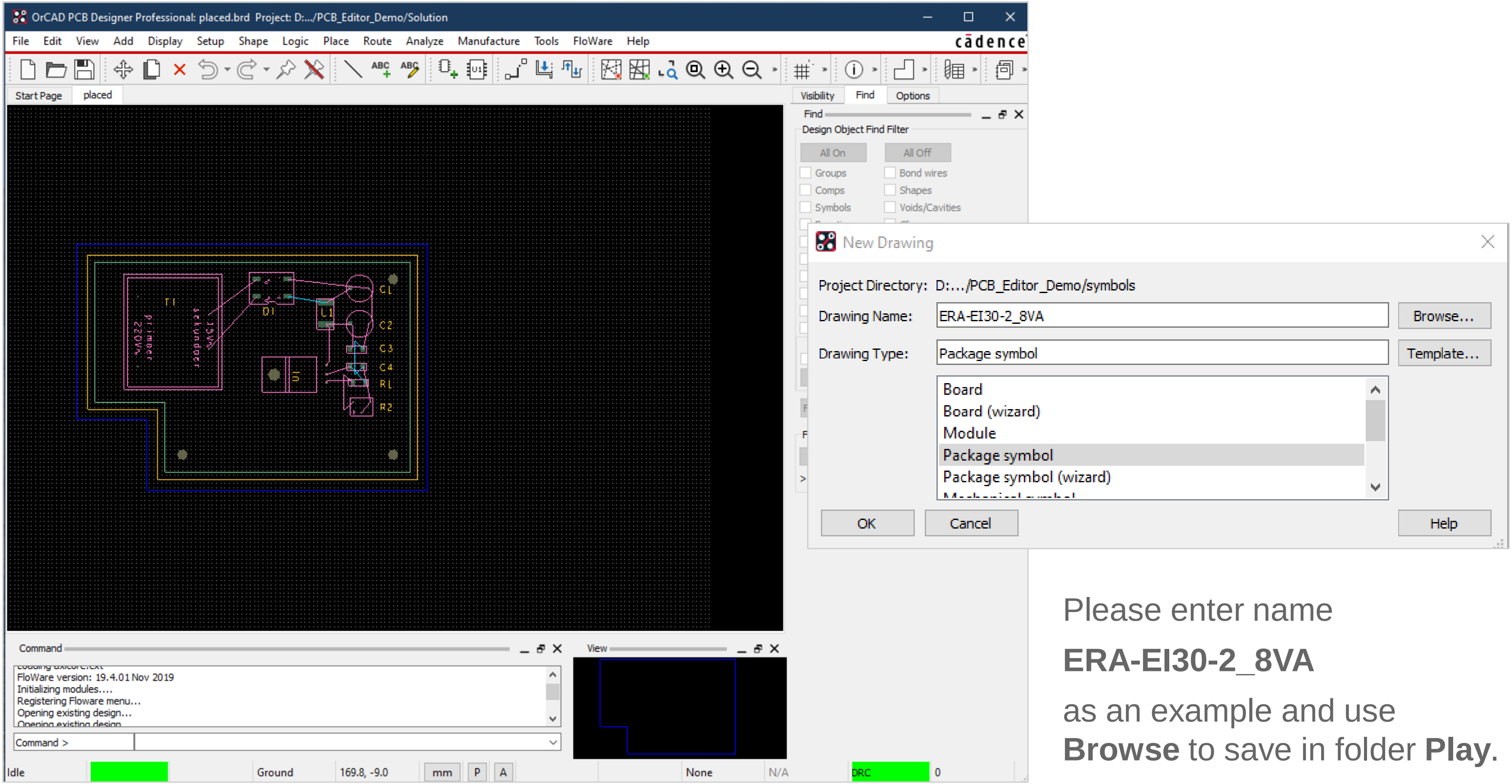
Creation of a new footprint includes following steps:

- Definition of units, decimal places and size of workspace
- Definition of origin. This is typically base point of component during placement
- Grid definition (does support easier curser positioning)
  Via keyboard any complex value can be entered
- Placement of pins (predefined padstacks)
- Assembly and silkscreen outline definition
- Definition of occupied component area for placement (Placebound_Top/Bot)
- Entry of possible height constraints of placebound_Top/Bot
- Addition or modification of component Text like REFDES or DEVICETYPE
- Saving in desired library

These steps are illustrated on next pages. Our example component is a transformer. Data sheet of our example component can be found in PCB_Editor_Demo folder and is named: **Datenbatt_Trafo.pdf.**

Lab: Symbol (Start)

Package Symbol Editor can be opened only from an already opened PCB Editor via **File New > Package Symbol**. In form you can enter name of a new footprint.



Please enter name

ERA-EI30-2_8VA

as an example and use **Browse** to save in folder **Play**.

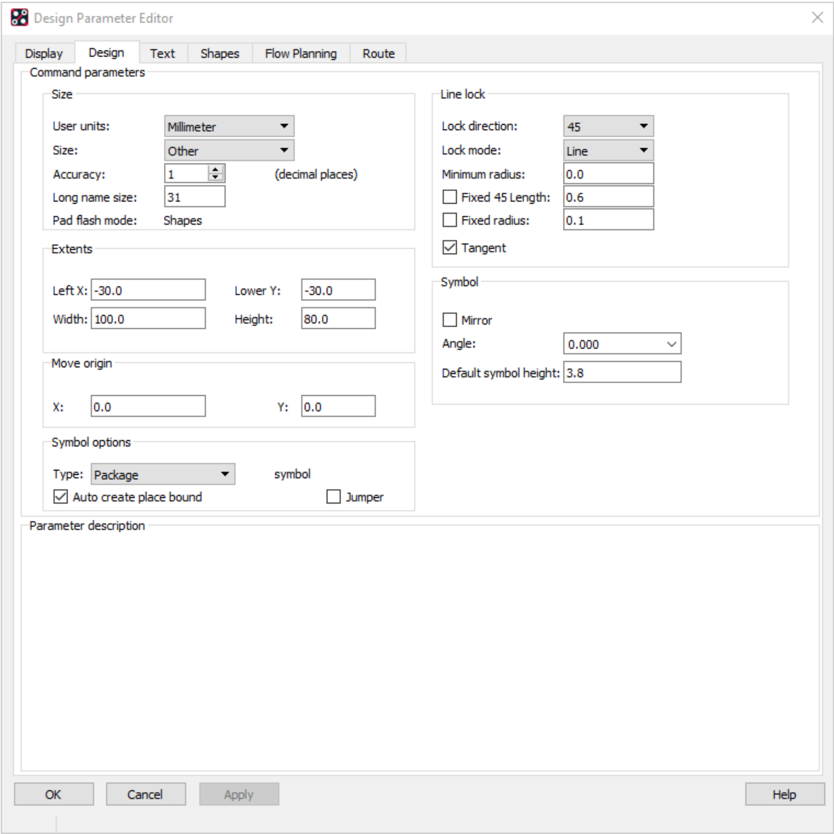
Lab: Symbol (Setup)

In **Setup > Design Parameters…> Design** you can set units, decimal places and size of workspaces well as origin.

Move Origin allows to move origin to a different location. Size of workspace stays untouched. Origin can be moved also with mouse via **Setup > Change Drawing Origin.**

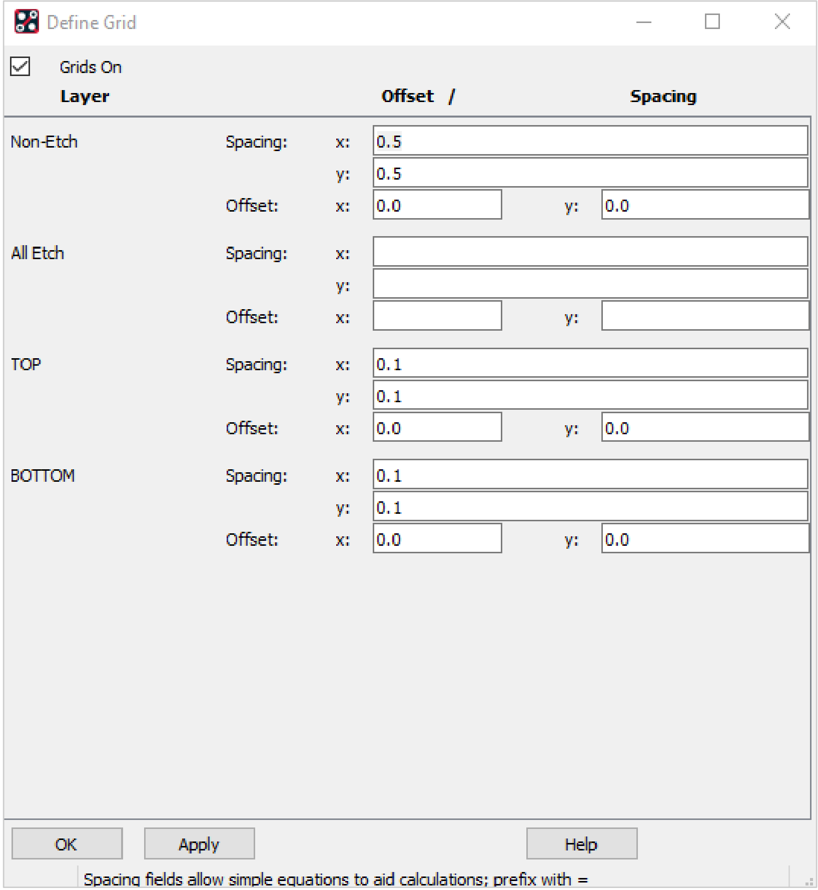
Please use following values for the example:

- Units: Millimeter
- Size: Other
- Accuracy: 1
- Left X: -30.0
- Lower Y: -30.0
- Width: 100.0
- Hight: 80.0



Lab: Symbol (Grids)

With **Setup > Grids…** you can define grid. For our example please use values from form on the right.



Tip

There are two separate grid definitions:

- **Non-Etch** for documentation layers
- **All Etch** for routing layers

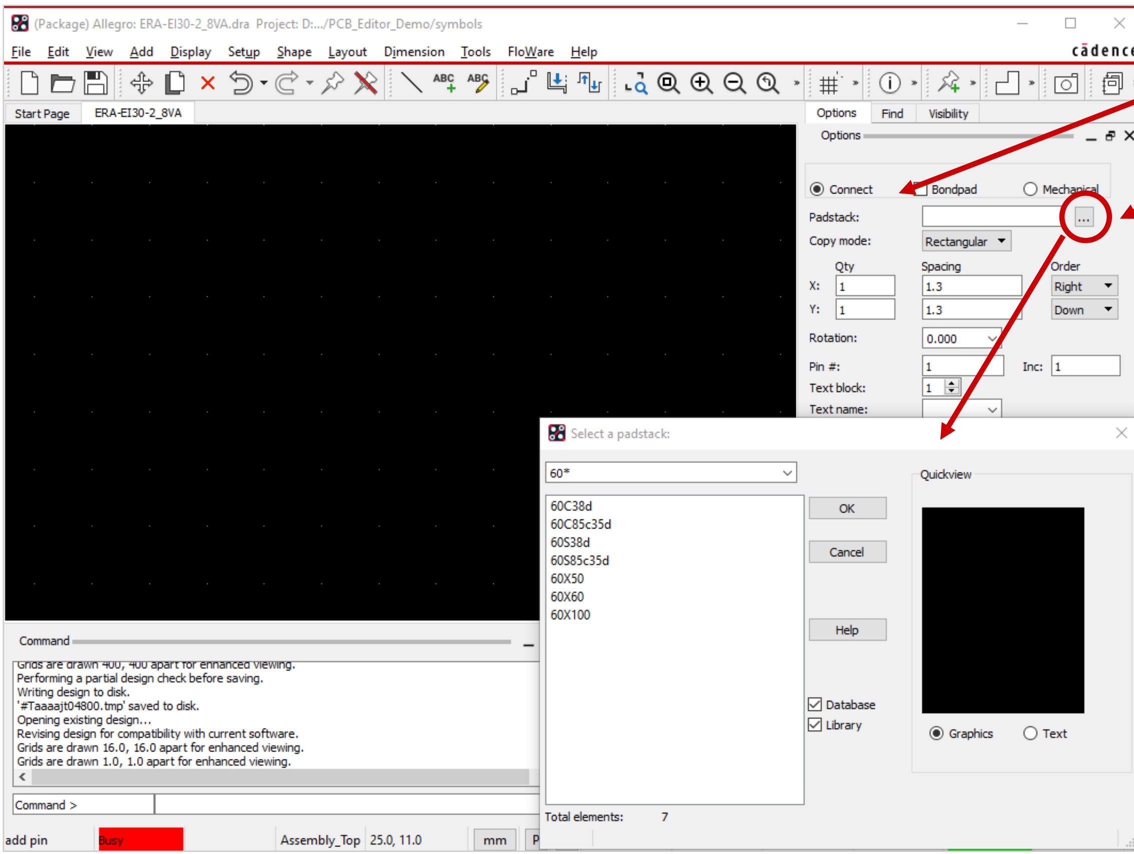
X and Y can be different.

Etch layers (top, bottom and other inner layers) can also use different grids.

Lab: Symbol (Pins I)

Please note that per default Symbol Editor implies top view.

1. Via **Layout > Pins** you can add pins.



All pins will be added as **Connect Pins** since they have an electrical function.

We choose padstack via appearing browser from preexisting library by clicking **OK** or a **double click**.

First pin will be placed on coordinates **X=0; Y=0**. This will be the origin and point for component placement in layout.

Fastest way is to enter coordinates in command line.

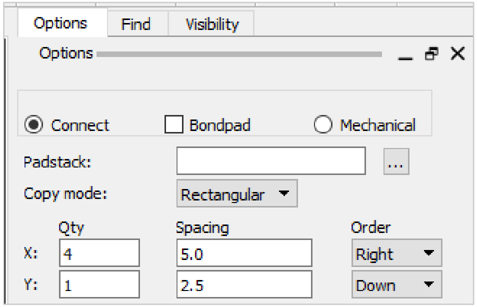
x 0 0 and **Enter.**

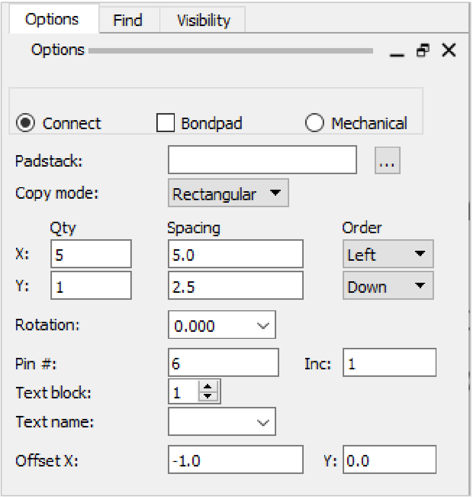
Alternatively entry can be done via **curser** (coordinates can be seen bottom right in control panel).

Lab: Symbol (Pins II)

Please follow instructions to place additional pins (2 to 5 and 6 to 10) as mentioned in data sheet.

2. **Add Pin** command is still active and pin 2 is on curser, ready for placement.
3. Now we would like to place pin 2 to 5 in one step. Double click on **Qty**-field **X** in option control-panel. Enter **4**.
4. Press **Tab** key and enter in field Spacing **5**, press again **Tab**. Option form is now ready to place an array of four pins. Spacing is 5,0 mm, first pin of the array is #2 and pins will be placed from left to right. Editor is waiting for position of array.
5. Please enter in command line **x 5 0** and press **Enter.**
6. To place next array (pin 6 to 10), enter in **Qty** field **X 5**. Form is ready for placement of next 5 pins. Spacing is still 5 and first pin is now #6. Direction is **left,** and editor is waiting for array position.
7. Please enter in command line **x 20 20** and press **Enter.**



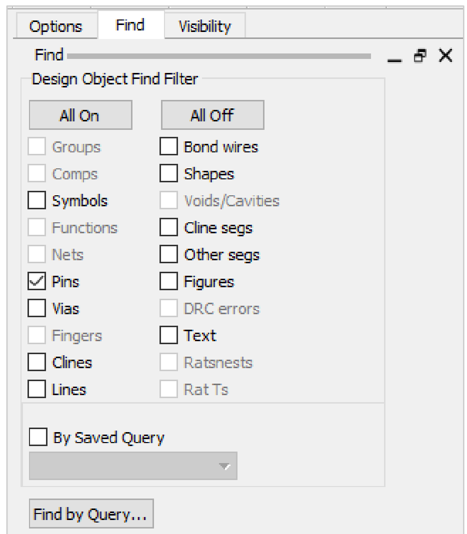


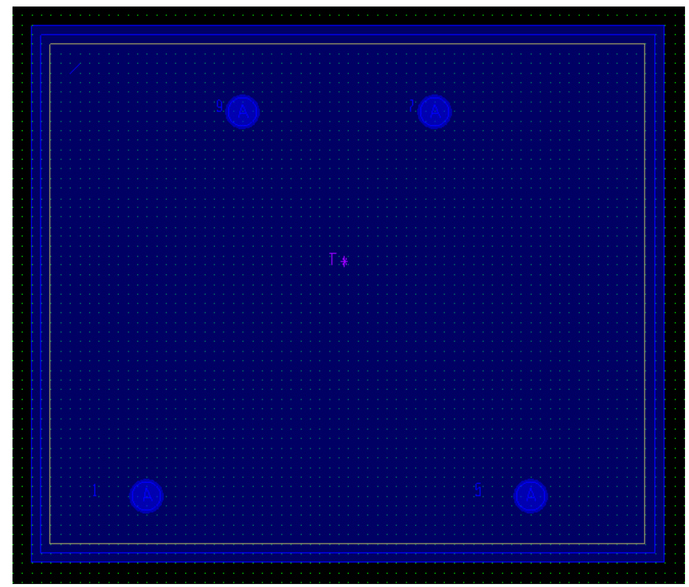
Lab: Symbol (Pins III)

Referencing to data sheet and schematic only pins 1, 5, 7, 9 are required for transformer. We need to delete non existing pins from footprint.

1. **Edit > Delete** o
2. Select find filter in control panel
3. **All Off** and **only pins** selected.
4. Delete unnecessary additional pins (2, 3, 4, 6, 8, 10)

   Please select above listed pins one after another.
5. With **RMB > Done** command will be finished.
6. Result see right.
7. **File > Save** (Overwrite = yes)
8. Package is stored in folder **Play** now.

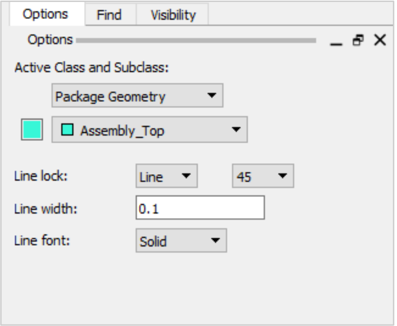




Lab: Symbol (Assembly Outline)

In next step we add an **Assembly Outline** typically used for assembly drawings. To make drawing task easier, grid should not be chosen to large or to small.

1. **Setup > Grid**, Define Grid window will appear.
2. Please enter in Non-Etch section for **x** and **y** e.g. **0.5**. Please note that values represent chosen units of editor. We work in mm now.
3. Click **OK.**
4. Zoom out to see entire work area for assembly outline.
5. Please choose **Add > Line** to draw a polygon. Please ensure that correct active layer is chosen in options window. Value for line width stands for line width in the later documentation. Line width 0 would not appear in output. Please enter a meaningful value, e. g. 0.1.

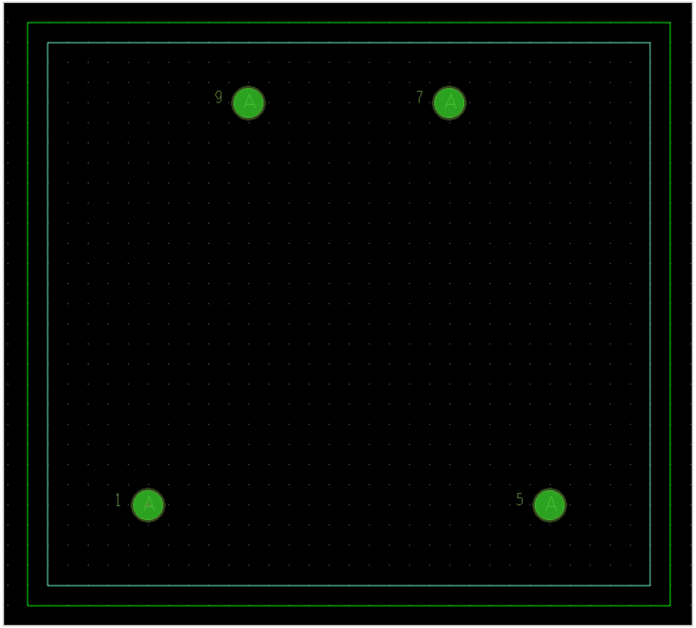


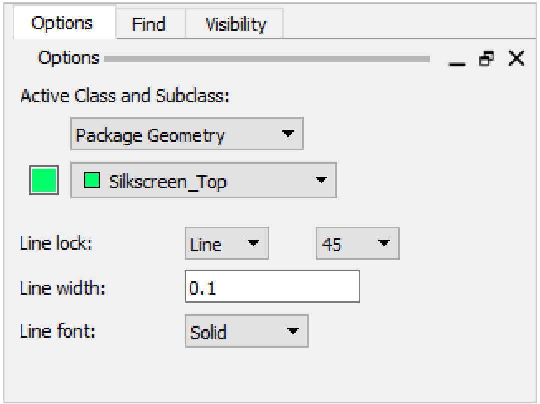
6. Draw a rectangle with dimensions X=33 und Y=28 (0.5 mm got added for assembly tolerance).
7. Click after another **LMB** on **0 0, 33 0, 33 28, 0 28, 0 0**
8. Click **RMB** and Done.
   You can enter values via command line like below.
   x 0 0 ENTER, **x 33 0** ENTER, **x 33 28** ENTER, etc.
9. Justify rectangle like illustrated in data sheet.
10. You can enter coordinates of polygon with real values related to origin too.

Lab: Symbol (Silkscreen Outline)

Next step we will add **Silkscreen Outline**.
We will reuse already defined grid.

1. Please make sure that active layer is set to Silkscreen_Top.
2. Please choose again **Add > Line** and line width = 0.1.
   Draw a polygon outside of assembly lines.



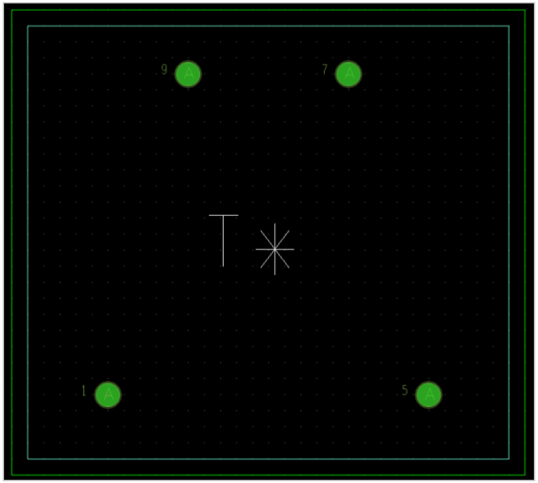


Lab: Symbol (Placeholder)

Label **REFDES** is required for every footprint.

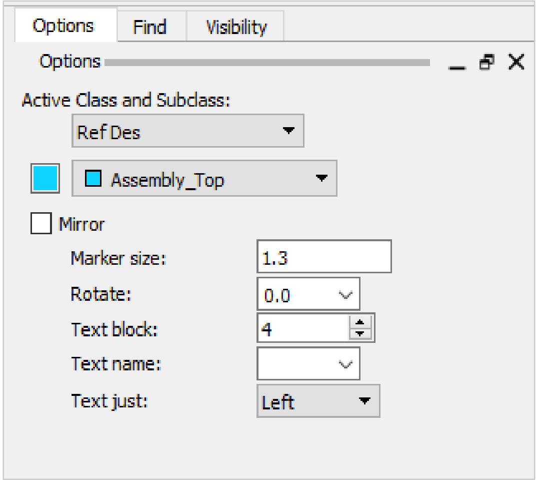
Definition of multiple placeholder for labels does make sense. They are used to show logic information. In PCB editor it is possible to define five labels max.

- REFDES (reference no. of component, R1, C2, etc.)
- DEVICE (device name from Packager)
- VALUE (value of component, e. g. 10 K for a resistor)
- TOLERANCE (tolerance if provided)
- PART NUMBER (part number for BOM)



1. **Layout > Labels > REFDES** or REFDES Icon R1

   Please notify settings for necessary layers.
   System will ask for desired position.
2. Click inside of assembly outline. System will ask for a text string.
3. Enter as an example a **T*** .
   This string will be replaced by real REFDES later.



Tip

Please note options for Marker Size, Rotate, Text Block, Text Just. They are important for size and adjustment of text string.

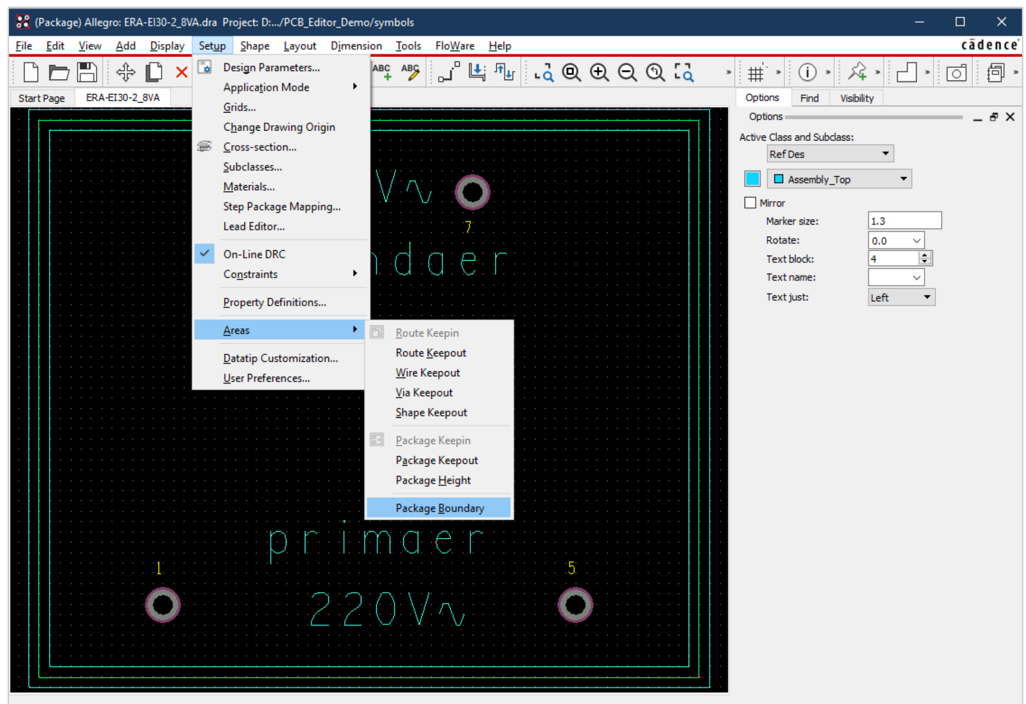
Lab: Symbol (Package Boundary)

Design Rule Check (DRC) is using package boundary to verify overlapping components and show an error. This also avoids placement of components in restricted areas (Keep Out Areas).

If you don't create package boundary yourself, system does generate one during save. This auto generated outline is not precise and does not represent real dimensions of component.

1. **Setup > Areas > Package Boundary** from main menu
2. Package_Geometry und Place_Bound_Top are automatically set.

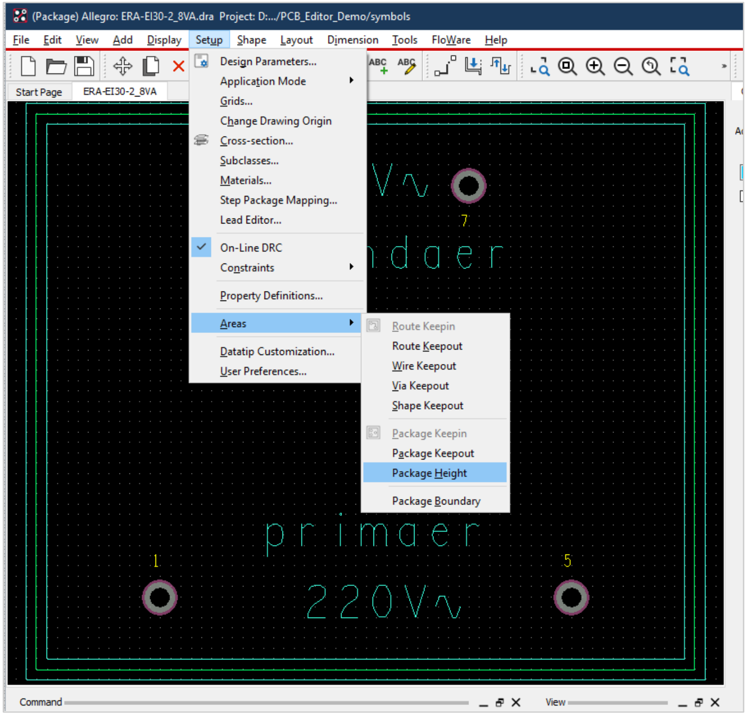
   Define size based on **worst case** from data sheet.
3. Click **LMB,** to enter polygon for placement boundary. To finalize polygon press **RMB > Done.** Polygon will be solid displayed.



Lab: Symbol (Package Height)

DRC program is using **Package Height** info to check component placement against predefined height restrictions. Package height is assigned to package boundary as **Max Height** or **Min Height**. It is not mandatory to define height information for every component. It is possible to define a default height in PCB Editor (Setup > Design Parameters… > Design > Symbol).

1. **Setup > Areas > Package Height** from main menu.
2. Select Package Boundary (filled polygon).
   Enter **29** in Max Height field.
   Min Height is not required.
   Height of transformer is 29 mm.
3. **RMB > Done,** to finalize command.
4. **File > Save.**
   System will save a **.dra**-file, and a **.psm**-file.
   .psm-file is used by Editor during placement.

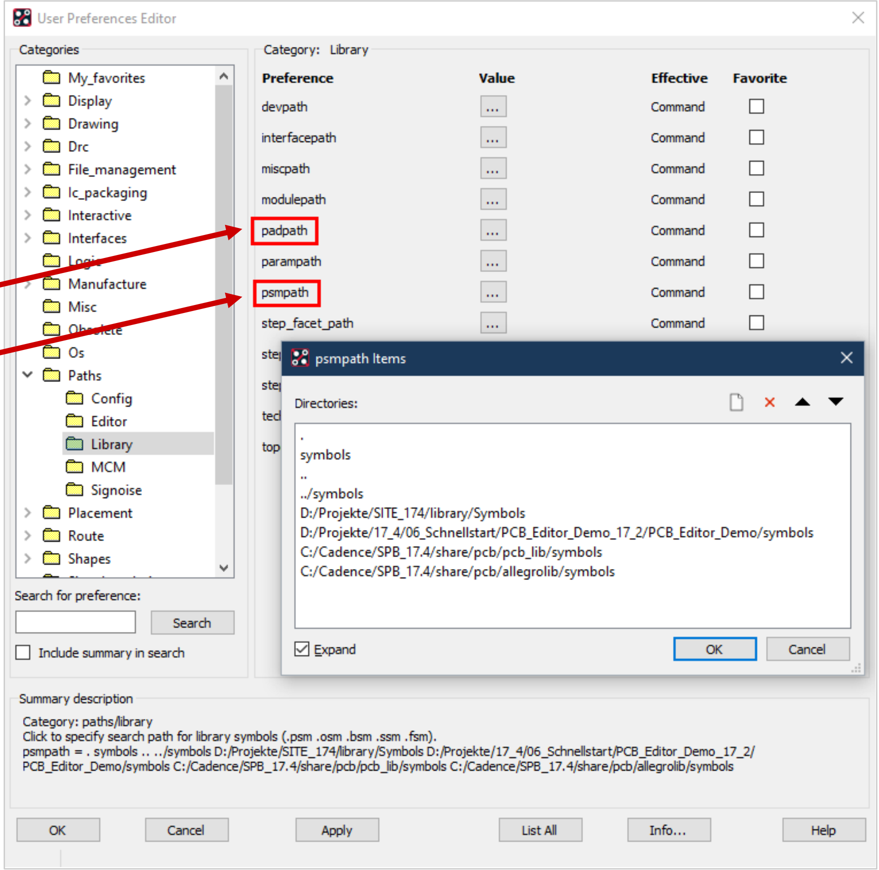


Save Location for Pads and Symbols

Files generated in Symbol Editor (.dra and .psm) as well as files generated by padstack editor (.pad) must be saved in psm and pad path. PCB editor need the files to find pads and symbols for placement. This will be referenced in user preferences too.

.pad-files

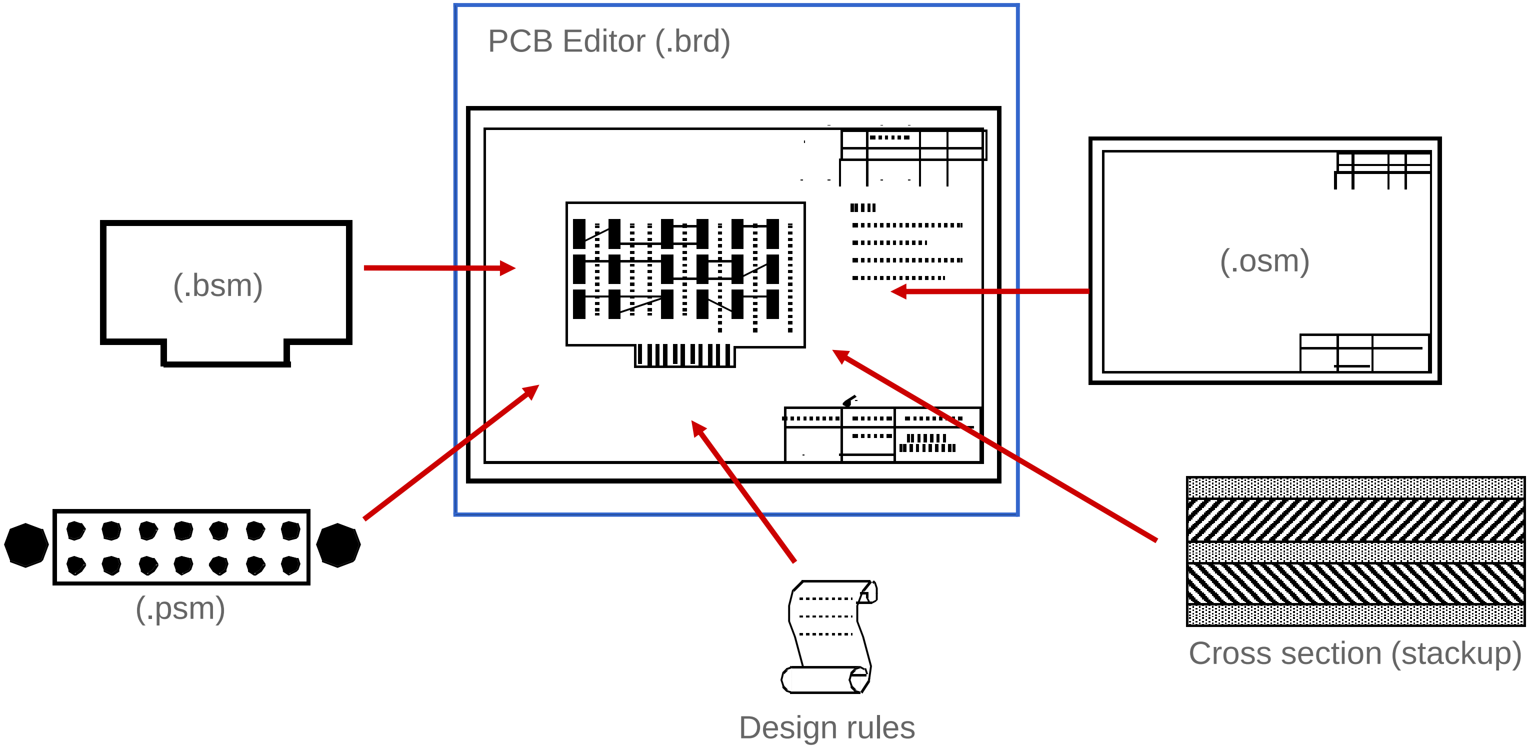
.psm and .dra-files



Board Setup

Board Overview

In a board all elements are used which were created by other editors. Image below shows an overview of elements used. How to create board templates will be described in a later capture.



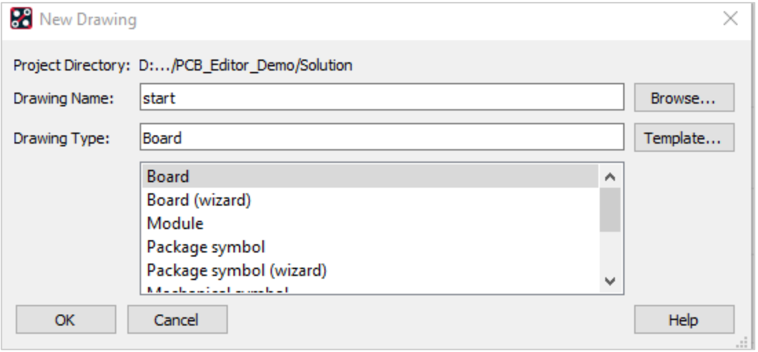
Tip

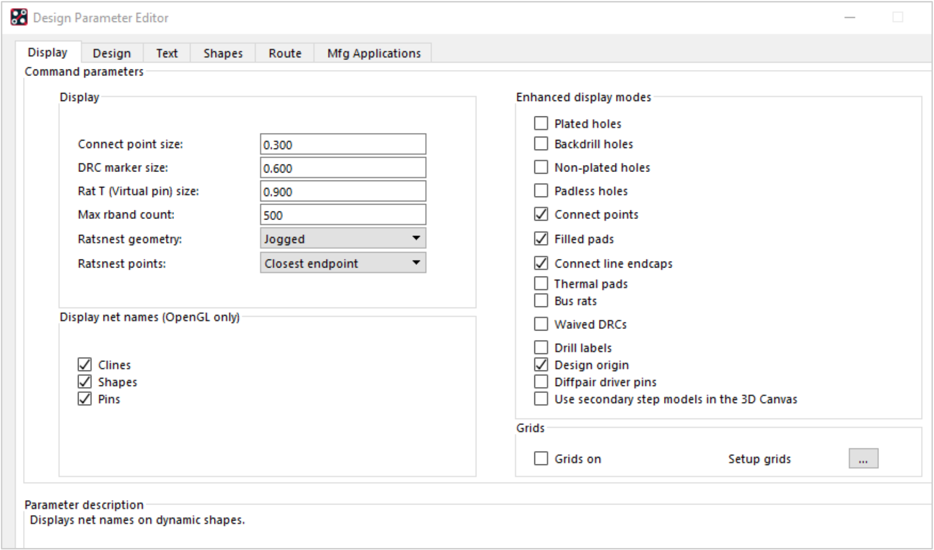
Different versions of labs are available in solution folder.

Lab: Board Setup

Following pages describe most important steps how to setup a board.

1. **File > New** from main menu
2. Please enter **master** into Drawing Name field.
3. Choose Drawing Type **Board.**
4. **OK**
5. **Setup > Design Parameters… > Design**
6. Please change values as shown in drawing parameter box on the left.
7. **OK**



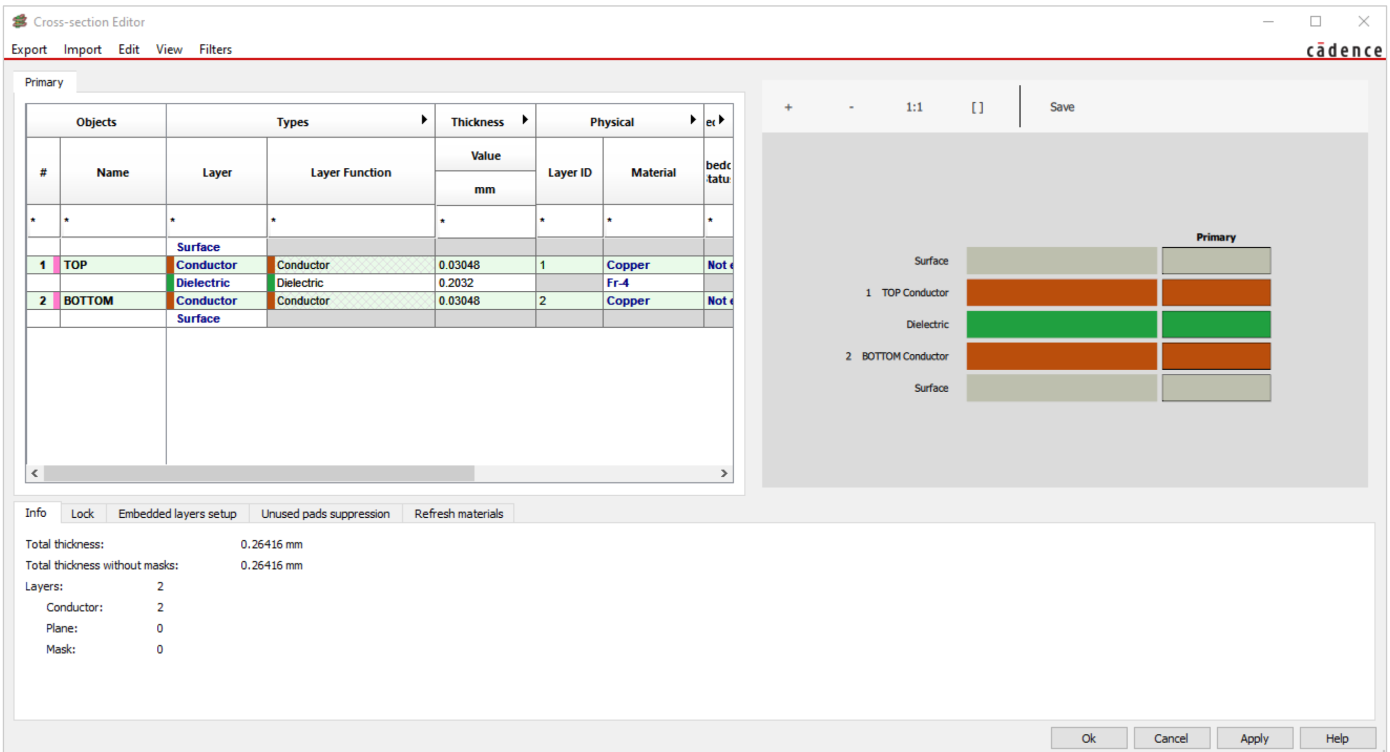


Tip

Drawing Extents should be large enough to include additional elements like drawing frame, title block. If there is not enough space, you will get an error message.

Lab: Layer Stackup

Layer stackup of board will be defined with Cross Section Editor. To open **Setup > Cross Section** or 
Via RMB on top of a existing layer you can add or remove layers.



Name of layers Top or BOTTOM cannot be changed. Additional layers can have any name.
Chosen names appear also in visibility window.

Lab: Board Outline

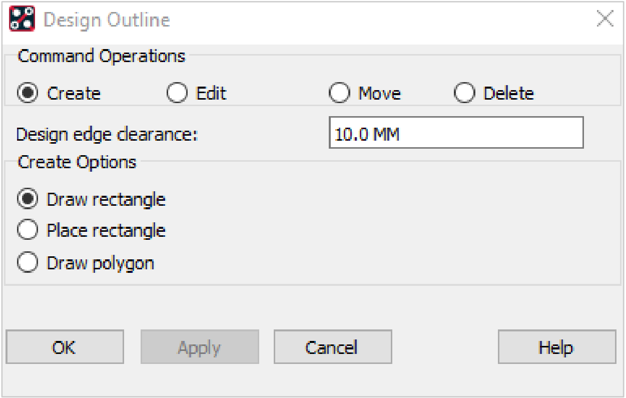
Via **Setup > Outline > Design Outline** contour of board can be defined.

Design edge clearance is an offset (smaller than board contour) to define clearance for Package keep in and Routing keep in.

Later keep ins can be modified manually.

On next pages we use a predefined board template.

If you don't want to create board outline yourself, you can reuse predefined one.

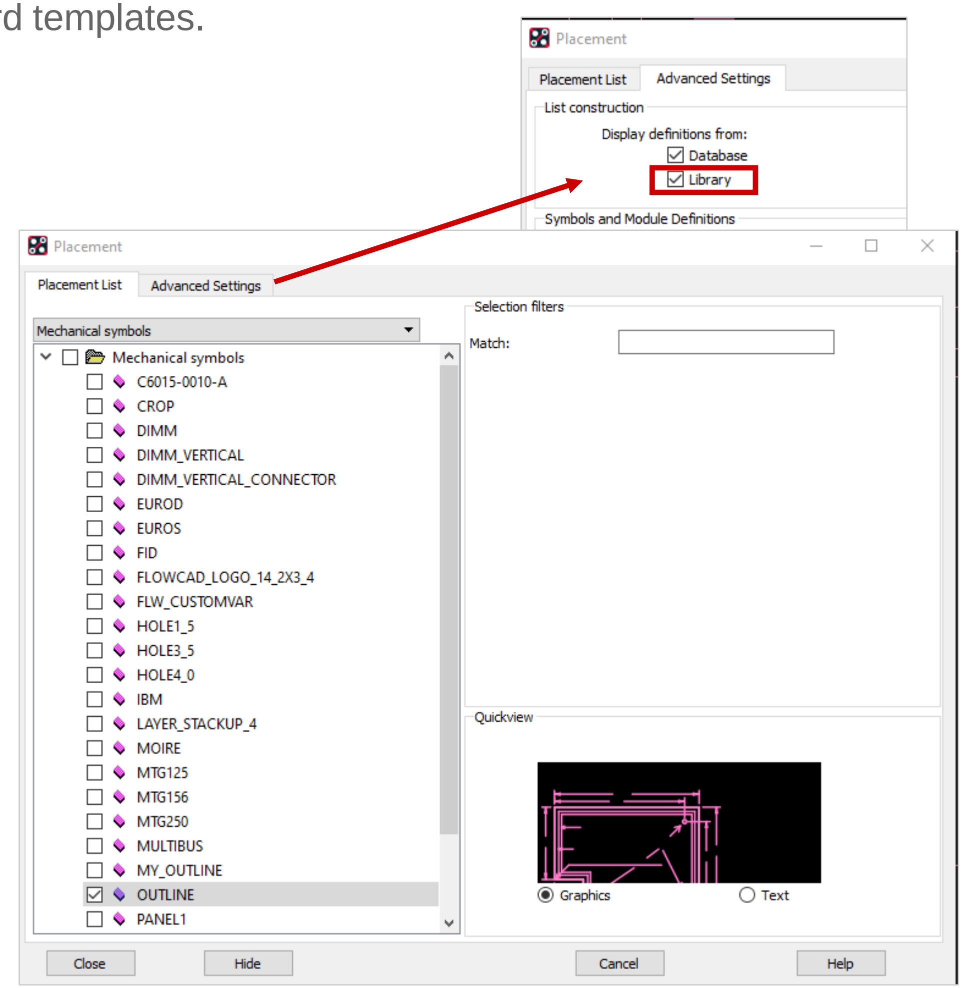


Lab: Board Symbol Placement

In this demo data set a board symbol is available. This board symbol can be used for next lab. In a later chapter we explain how to create board templates.

Load board symbol into still open master board **start.brd** as described below.

1. **Place > Manually** from main menu. Placement Box will appear.
2. In **Advanced Setting** tab select both options, **Database** and **Library.**
3. In Placement list expand mechanical symbols and select **Outline** (or yourself defined symbol).
4. Type in command line **x 0 0** and press **Enter.**
5. **RMB > Done.** Mechanical symbol is placed.
6. **File > Save As.** **master.brd** file will be saved.
7. Please **do not** close PCB Editor yet.



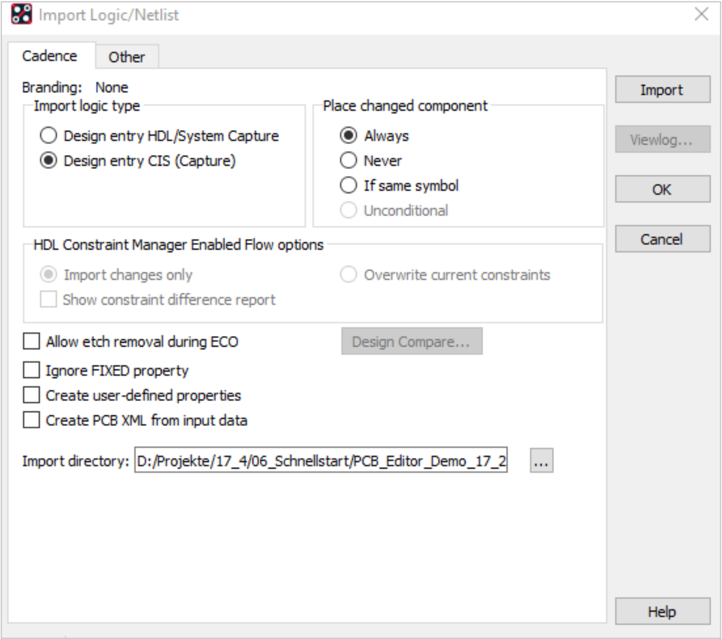
Import of Schematic Data

Lab: Import of Logic Data

This chapter explains how to import schematic data into PCB Editor. Some important information regarding next lab:

- Schematic is complete.
- Necessary transfer data as well as schematic are available as sample data in folder **Project2.** Information regarding transfer data can be found in appendix.

1. Please load logic data into **start.brd** file via **File > Import > Logic.**
2. Select import logic type **Design Entry CIS**.
3. Configure correct import folder **<Path>\PCB_Editor_Demo\project2.**
4. **Import Cadence** (maybe an incorrect import is reported).
5. **File > Save As… (netlist.brd) Do not close!!**



Design Constraints

Design Rules

Before we start to place components, we should define some basic rules to control placement. In addition we will define some necessary design rules for routing task. Availability of rules and definition options are driven by used license.

All design rules are managed in Constraint Manager. They are divided in categories below:

- Electrical Rules: Design rule, to categorize electrical attributes like impedance, topology, …
- Physical Rules: Definition of trace width, vias, differential pairs, …
- Spacing Rules: Clearance rules between design objects like traces, pads, vias, copper areas, …
- Same Net Rules: Clearance rules between objects related to same net
- Manufacturing: PCB manufacturing rules like component clearance, mask clearance or minimum copper rings

Rules are separated into two levels:

- Standard Rules: Always available (default) rule set. This rule set is used for majority of nets. Typically for all nets without a specific rule requirement.
- Special Rules: User defined rules sets. These rules are different from standard definition and get assigned only to specific nets, like power or critical signals.

Next to design rules there are assigned properties and DRC violations listed.

FlowCAD

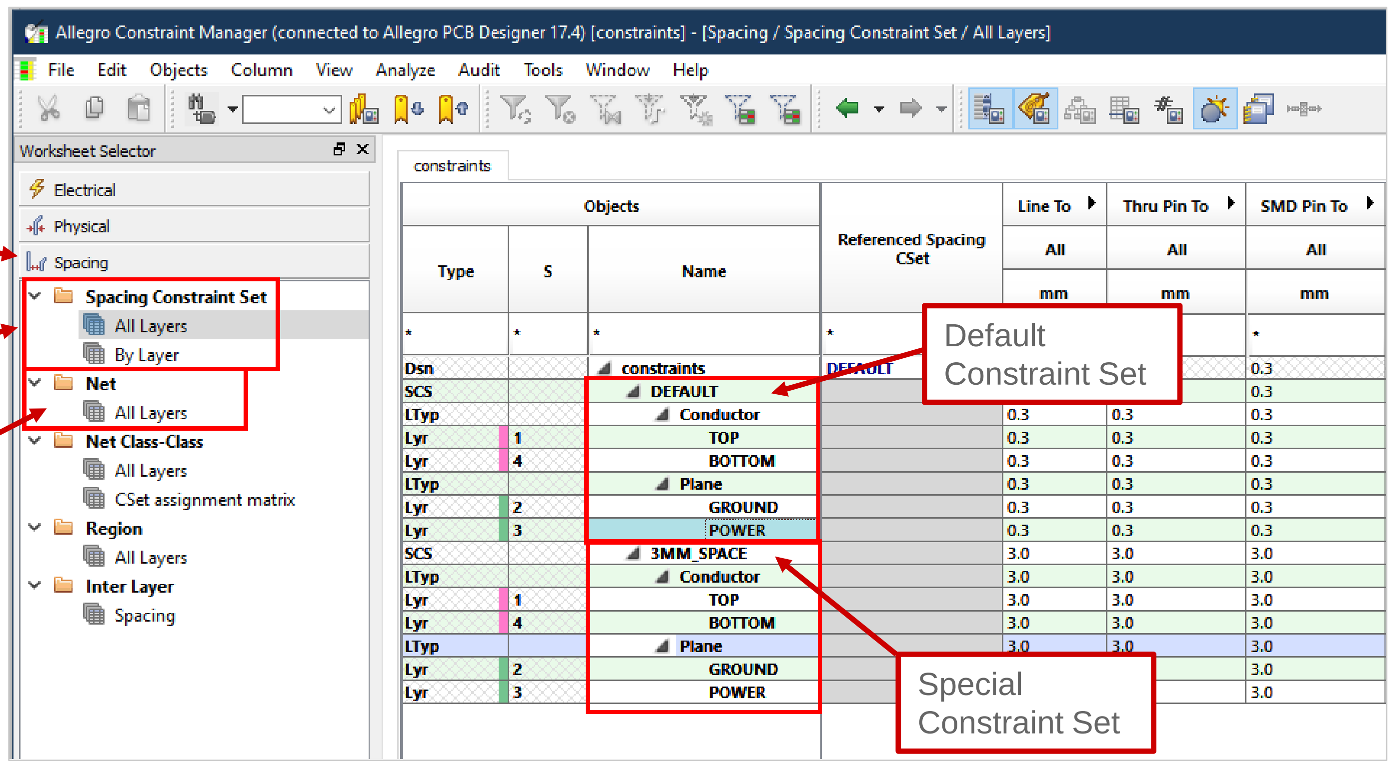
Constraint Manager Overview

Start Constraint Manager via:
Setup > Constraints > Constraint Manager… or 

Constraint Domains

Constraint Sets

Constraint Assignment



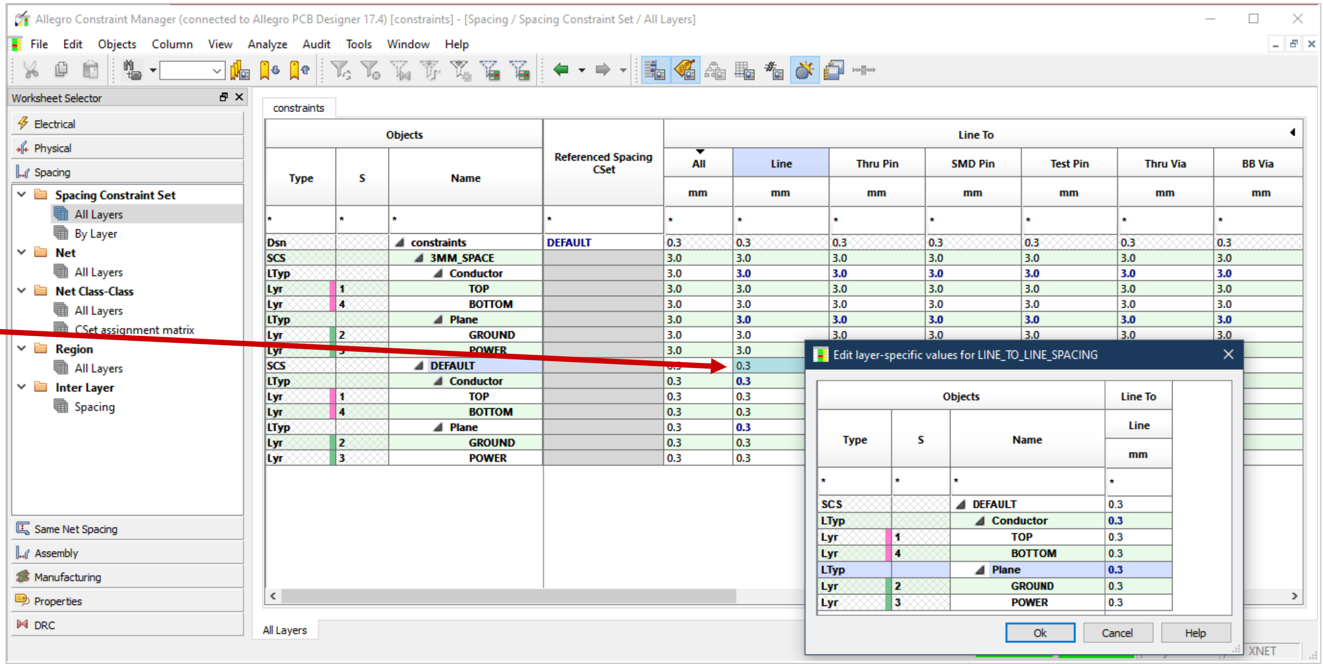
Standard (Default) Design Rules

Standard rules should be entered in Cset **default.** Individual objects can have different values to each other. All nets without a separate assignment from an additional Cset or a direct entry will be checked against **default** Cset.

Tip

If there is only one single value in an object to object definition, it is valid for all layers together.

Via **RMB > Change** within a Cset it is possible to assign for this cell different values for each layer.



Special Design Rules

If you work on a more complex design, you need to assign separate rules to dedicated nets which differ from default values. Some nets need modified spacing to each other.

This does require usage of Extended Design Rules. Please note next necessary steps for spacing and physical rules.

- **Step 1:** Creation of a new Constraint Set (CSet).
- **Step 2:** Creation of net classes, where same constraints apply.
- **Step 3:** Assignment of Constraint Sets or Net Classes.

Lab

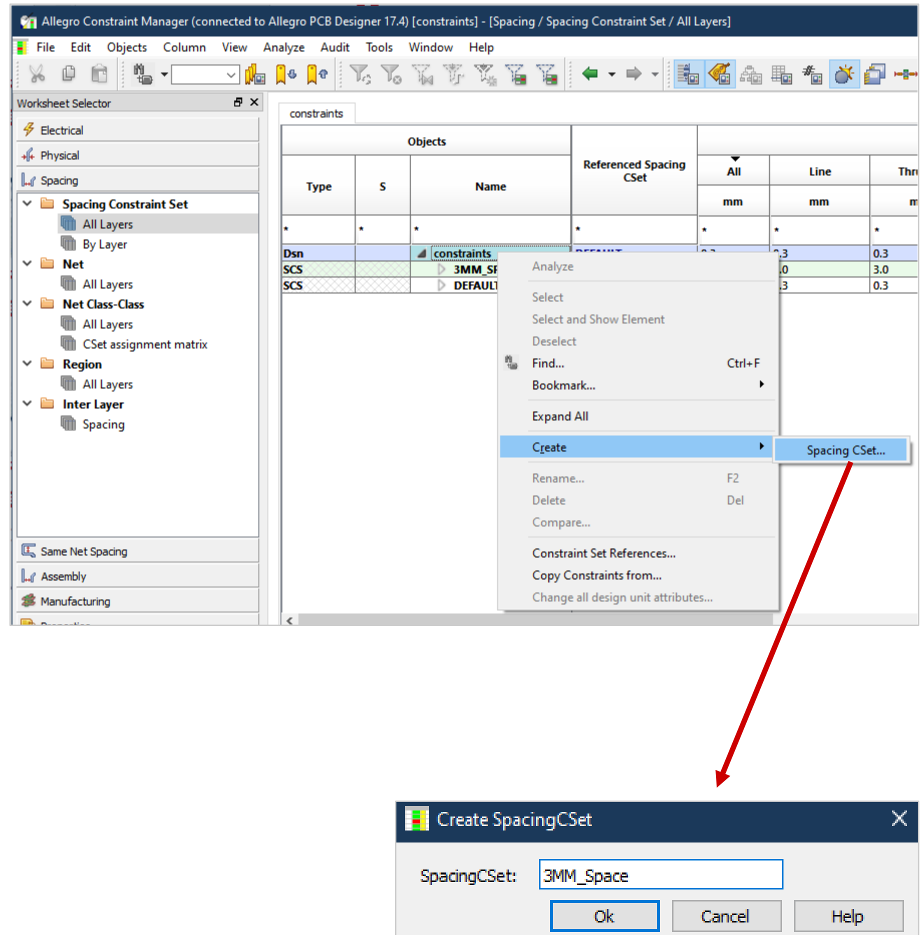
Next pages will guide you through essential steps in next lab. We will use **netlist.brd** file.

Lab: Step 1 – New Rule Set (I)

Creation of a new rule set (CSet)

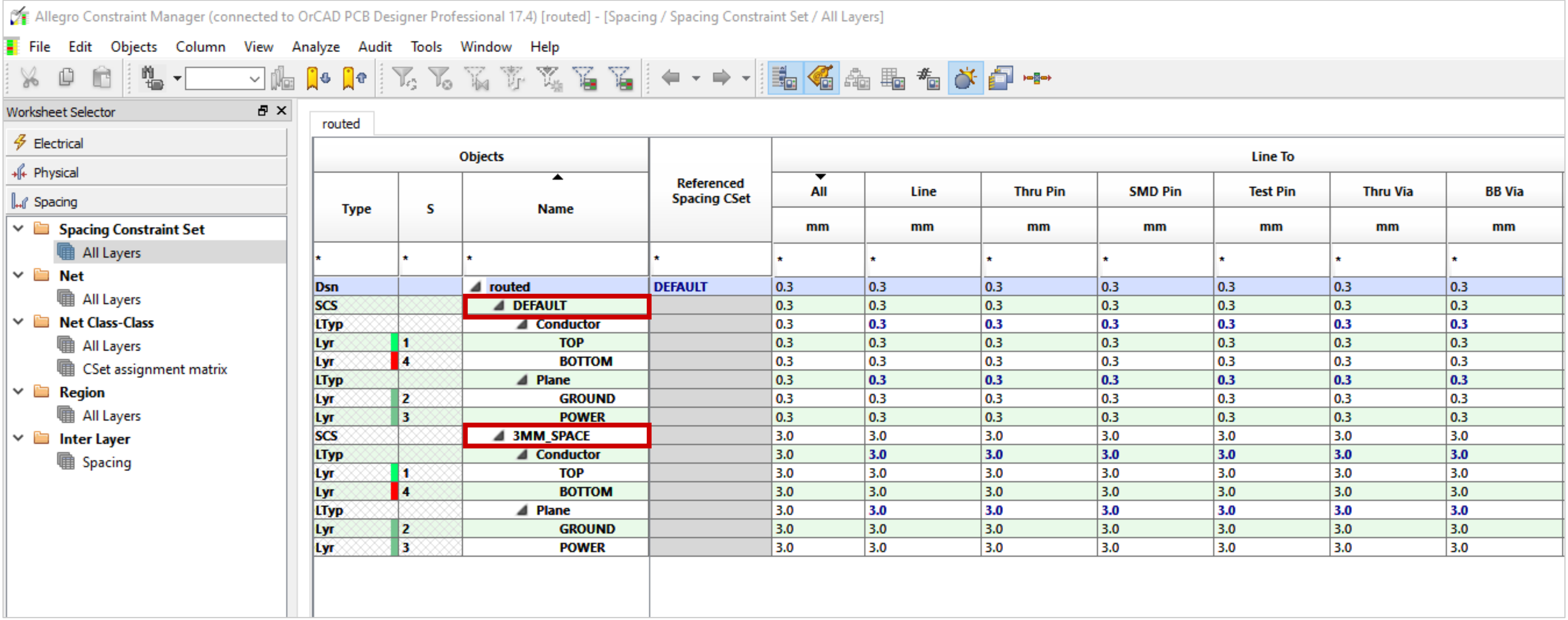
If possible, all rules should be organized in **Csets.** This will simplify the assignment to multiple nets. Each rule can be assigned individually too.

1. Creation of a new **Spacing CSet:** Select section **Spacing > RMB** on **Design Name** click **> Create > Spacing CSet…**
2. Enter name **3mm_Space** and click **> OK.**



Lab: Step 1 – New Rule Set (II)

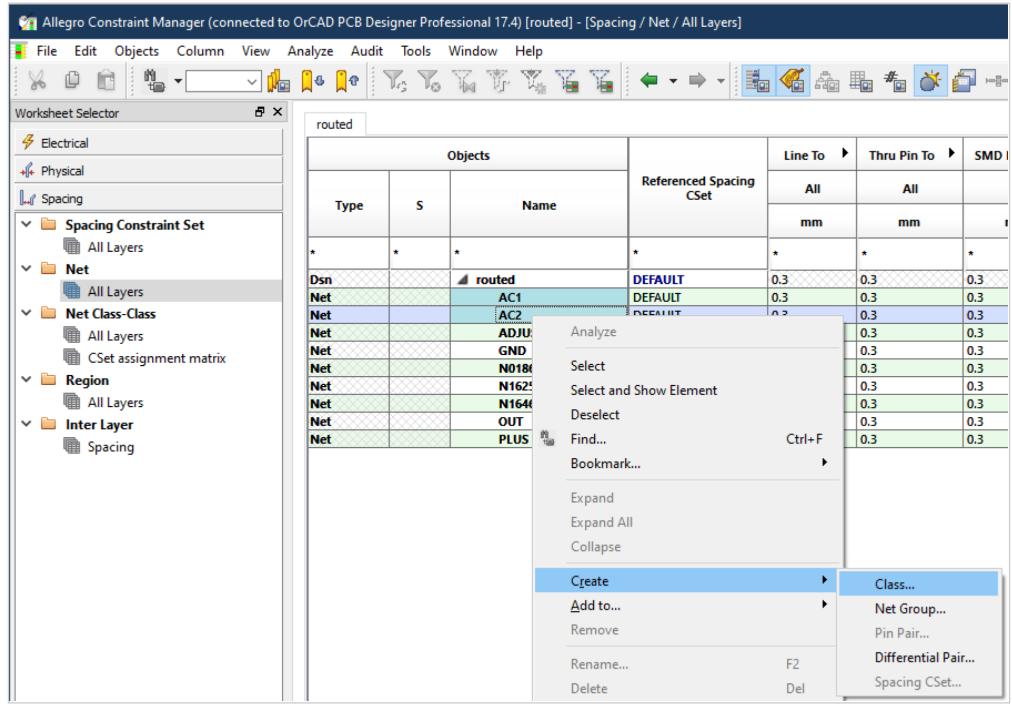
3. Please choose **Spacing Constraint Set > All Layers > All.**
4. Change all values of **DEFAULT** rule to **0.3 mm.**
5. Change all values of **3MM_SPACE** rule to **3 mm.**

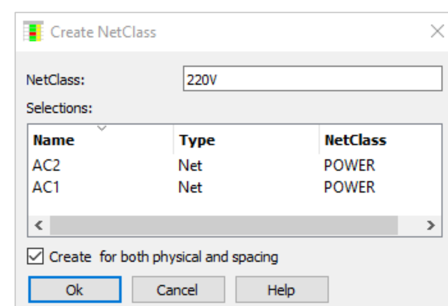


Lab: Step 2 – Net Class (I)

Creation of a net class

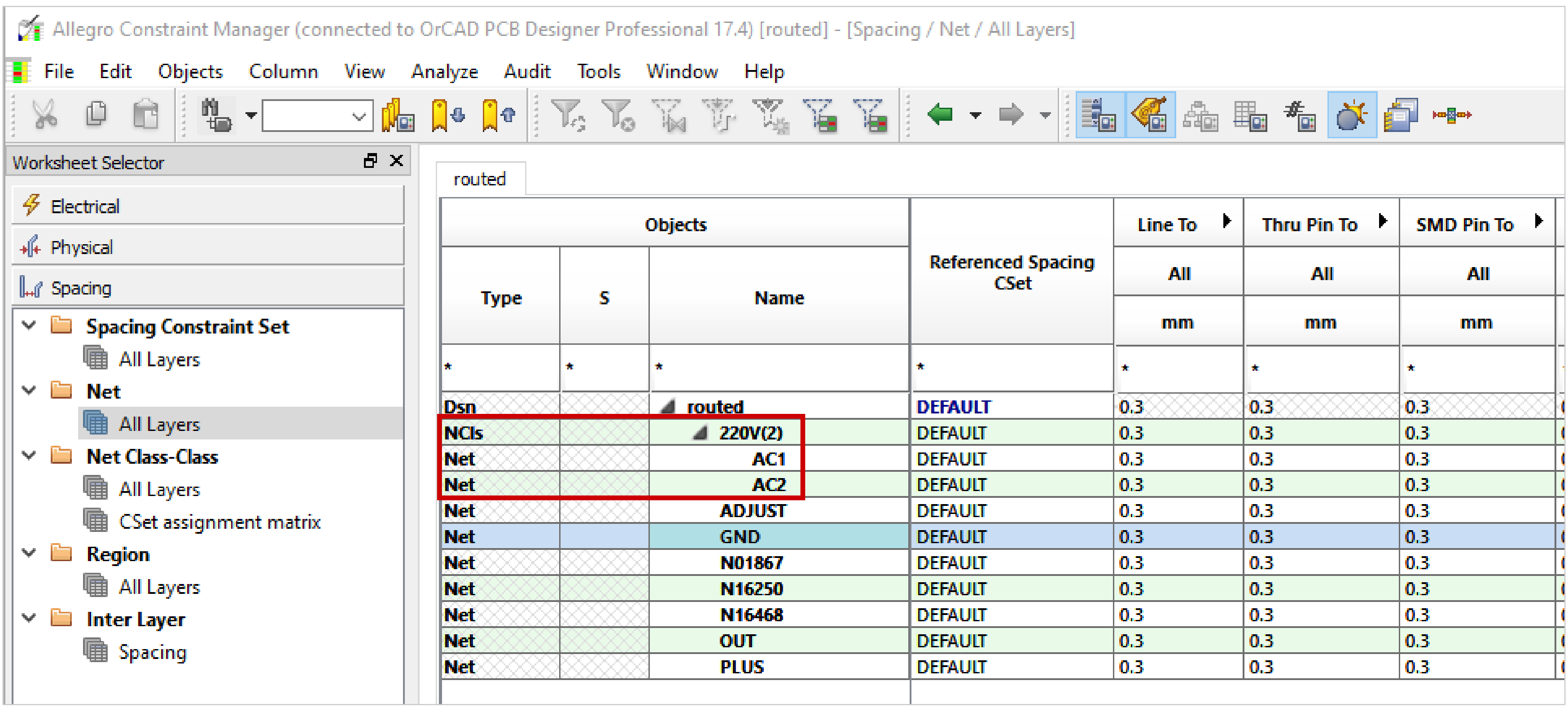
1. Select nets AC1 and AC2 and click **RMB > Create > Net Class… >**
2. Enter name **220V** and click **OK.**





Lab: Step 2 – Net Class (II)

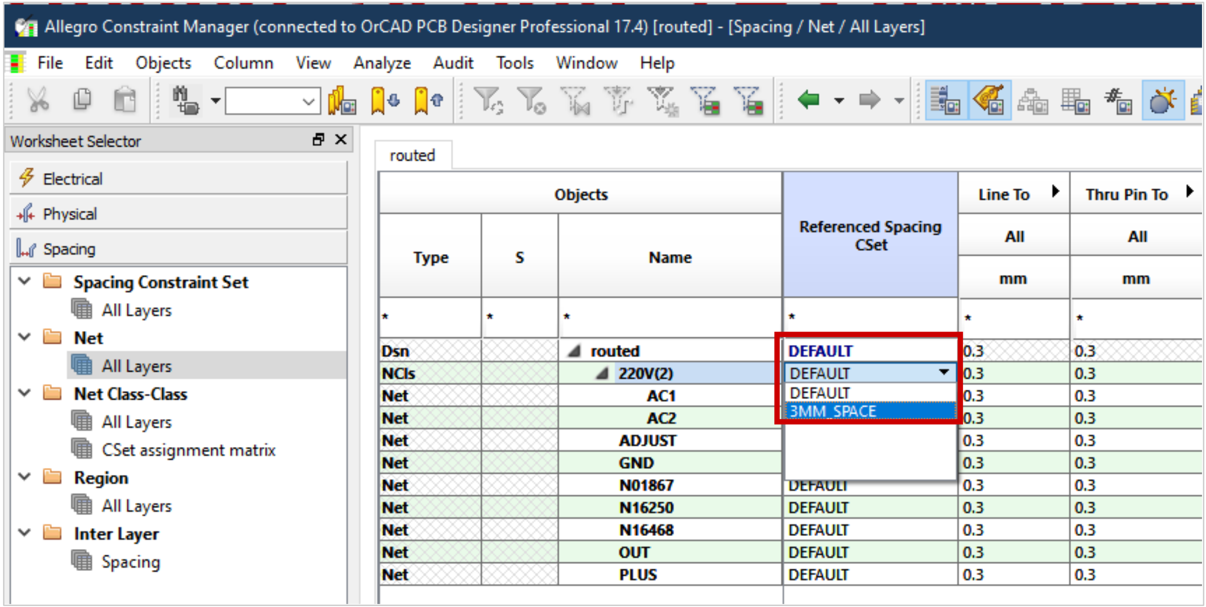
4. Now nets **AC1** and **AC2** are members of **Net Class 220V.**



Lab: Step 3 – Assignment

Assignment of rule sets to net classes

1. Chose **Spacing > Net All Layers > Line**.
2. Select field on the right next to net class name **220V.** A selection window will open.
3. Select **3MM_SPACE.**
4. Now net class 220V is assigned to rule set 3MM_SPACE.



Physical and Same Net Rule Sets

Handling of rule sets in **Physical Worksheet** and in **Same Net Spacing Worksheet** are identical to steps 1 to 3 of **Spacing Worksheet** in previous lab.

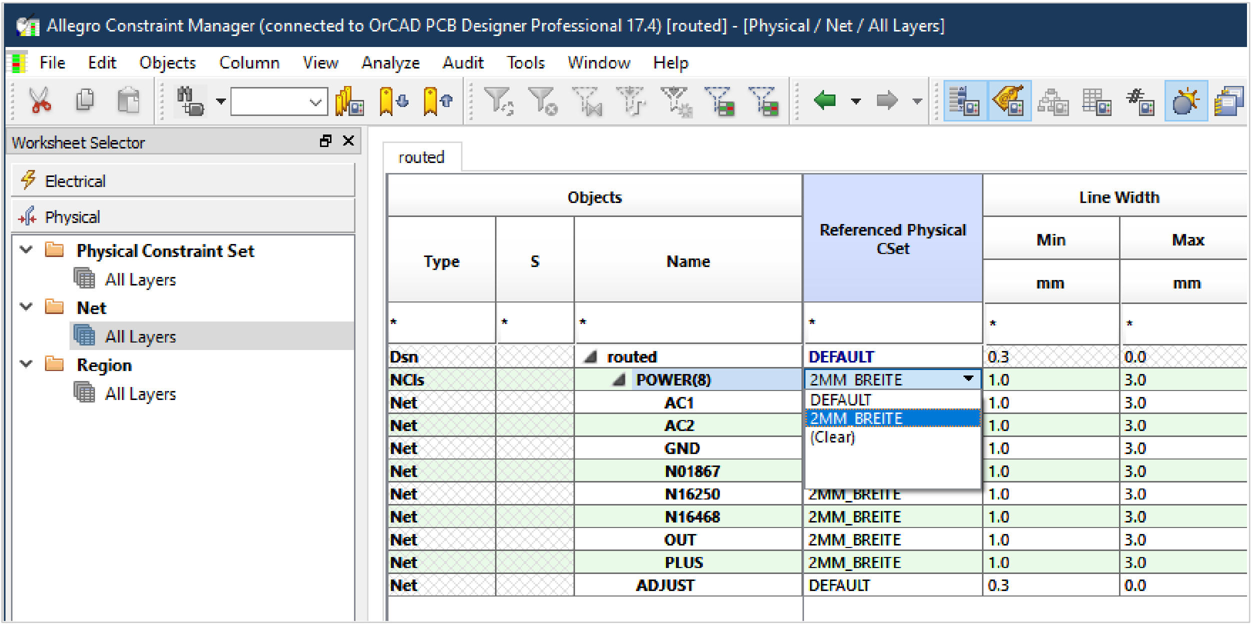
You just need to select desired category upfront in **Worksheet Selector** of CM.

Tip

Rule sets of worksheets

- **Physical**
- **Spacing**
- **Same Net Spacing**

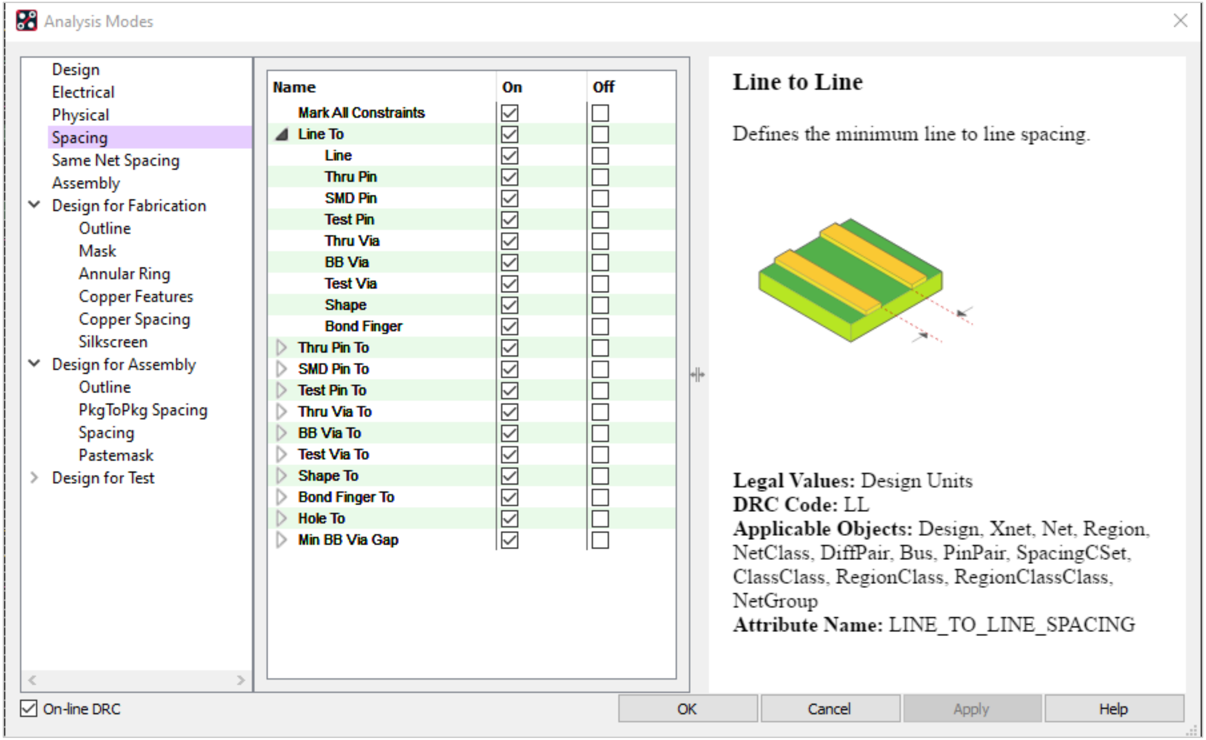
are completely independent and must be created separately and assigned separately.



Constraint Modes 1 (I)

Defined and assigned rules in Constraint Manager must be activated.

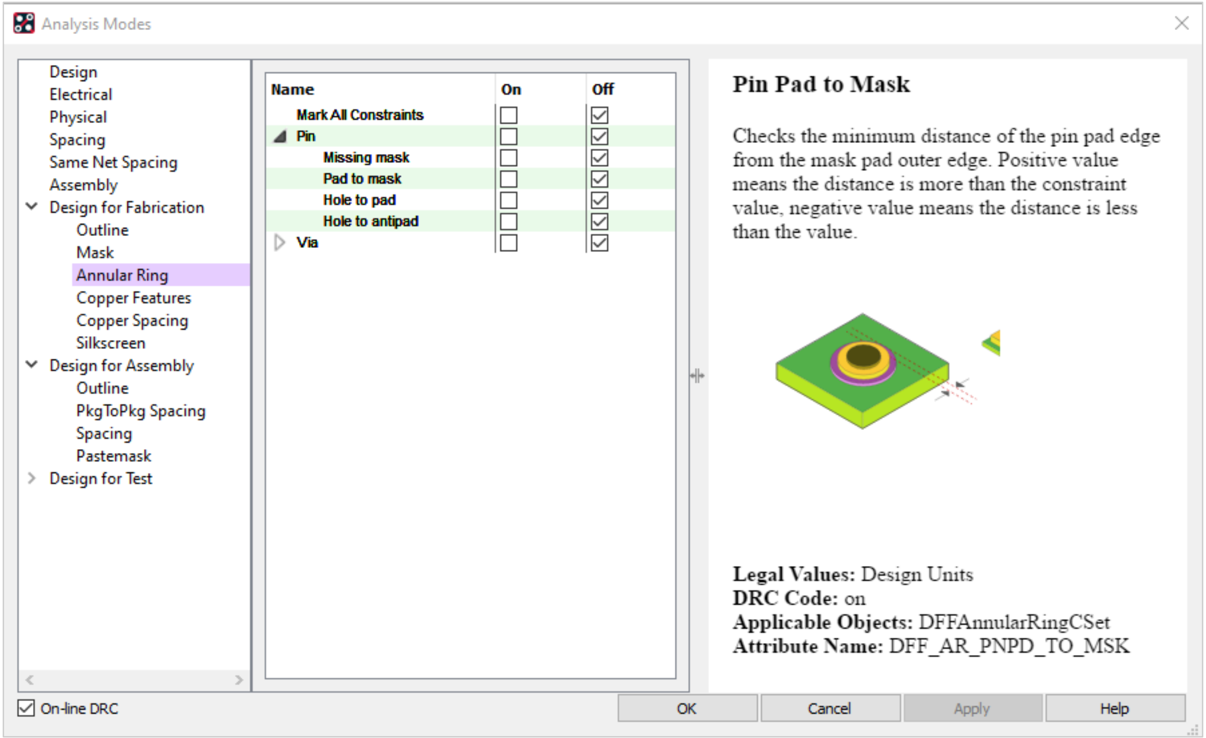
This can be done in Constraint Manager under **Analyze > Analysis Modes** or in PCB Editor under **Setup > Constraint > Modes**



Constraint Modes 1 (II)

Constraint Manager allows to configure many additional checks. In this quick start document we don't take it into consideration.

In **Analysis Mode** you can find many notes to different checks.



Part Placement

Part Placement (In General)

After setup of Design Constraints we focus on multiple options for part placement in this chapter. Lab is based on our known example.

PCB Editor needs data below for part placement:

- Netlist
- Package Symbol (footprint)
- Padstack
- Shape definitions (for specific pad geometries)

Footprints and padstacks (including shapes) must be available for PCB Editor via libraries.

Paths to libraries are defined via **PSMPATH** and **PADPATH** and stored in **env** file.

Complementary elements for placement are:

- Floor planning (can be pre-defined in schematic using ROOM property).
- Package keep outs (prevents to place components in restricted areas).
  Can be defined in **Setup > Areas > Package Keep out.**

Part Placement (Type)

We differentiate three types of placement:

1. Manuel placement (**Place > Manually**): From list of unplaced components, components get selected and manually placed.
2. Quickplace (**Place > Quickplace**): Semi-automatic placement of component groups based on selected criteria.
3. Auto placement (**Place > Autoplace**): Automated placement of components. There are additional definitions required, e. g. placement grid.

We will focus on manual placement and quickplace only.

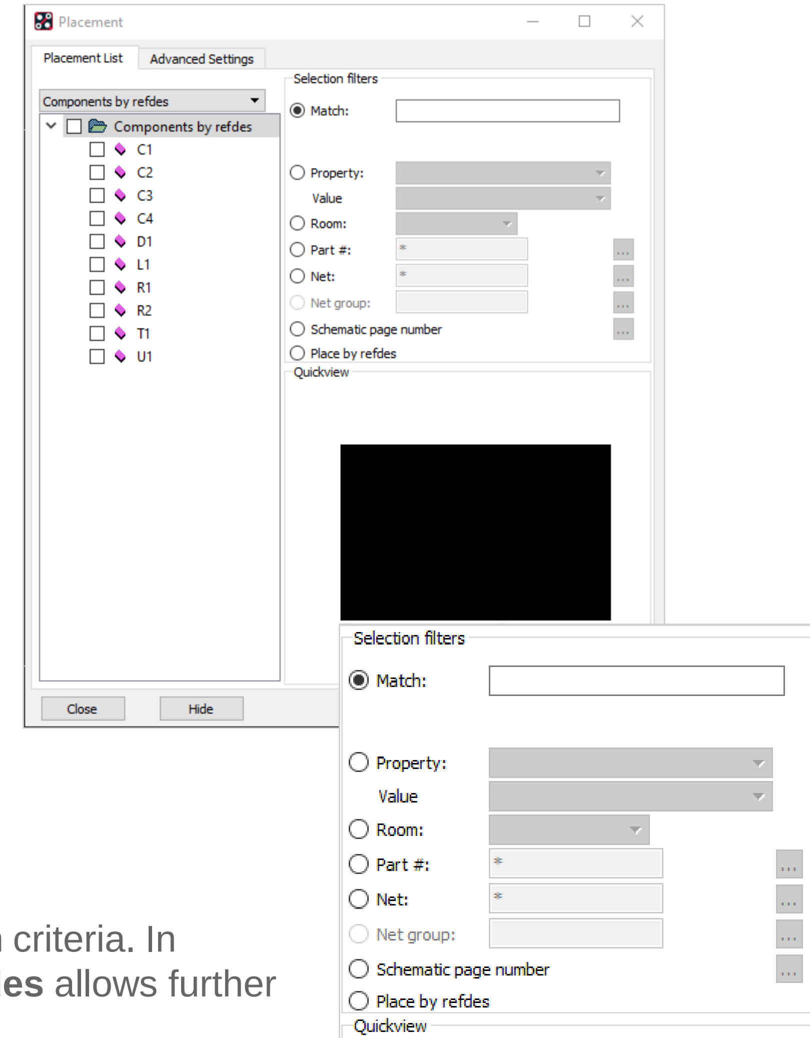
Especially for manual placement, placement grid (**Setup > Grids**) has to be set appropriately.

Place Manually…

Start manual placement with **Place > Manually** or

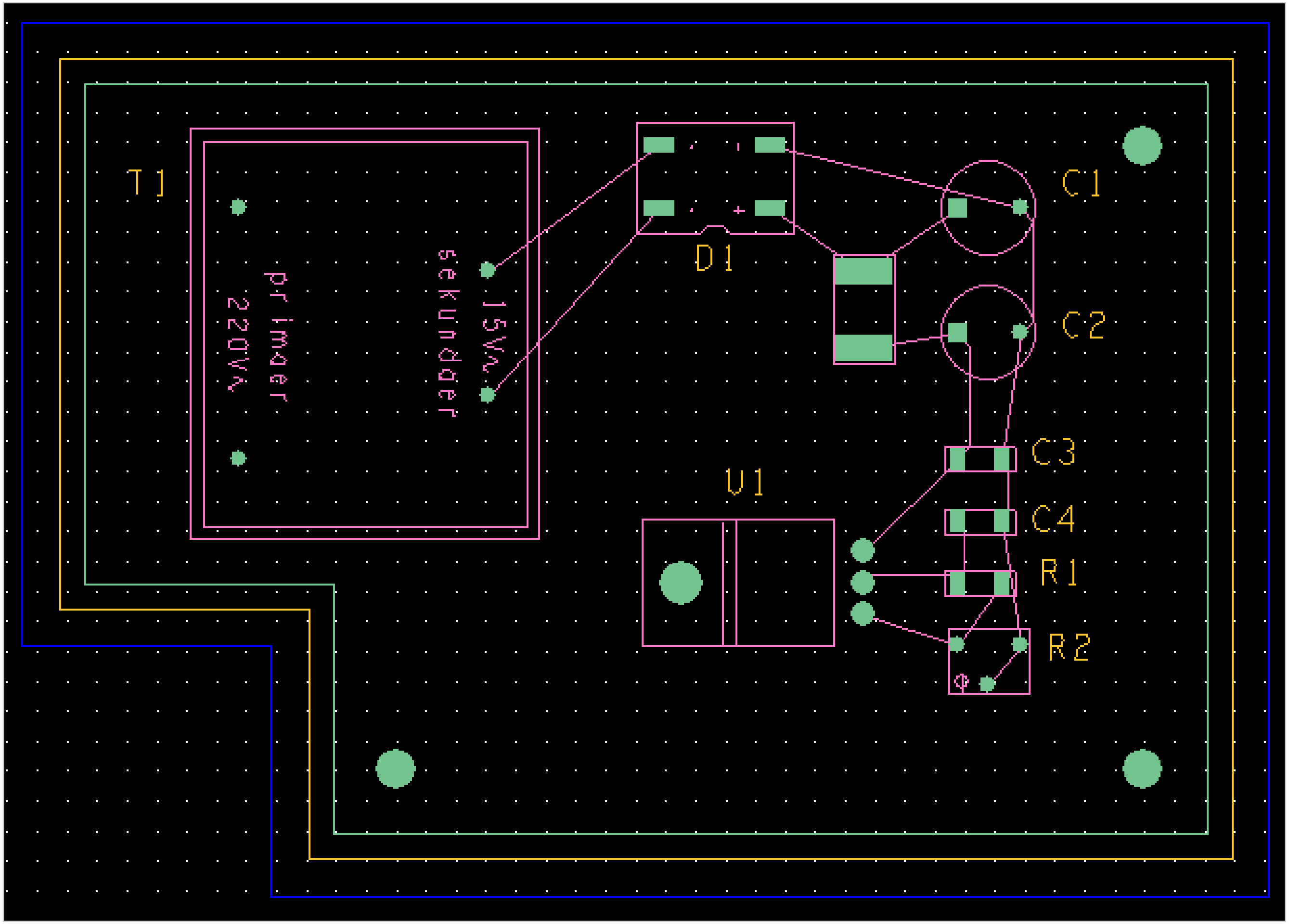
There a **five** options available:

- **Components by refdes**
  Select components by REFDES, based on loaded netlist
- **Components by net group**
  Based on net group definition in Constraint Manager
- **Package Symbols**
  Placement of packages (footprints) without considering electrical information or netlist
- **Mechanical Symbols**
  Placement of mechanical elements like mounting holes or additional outline data
- **Format Symbols**
  E. g. drawing frame for documentation



Selections Filter provides a variety of practical selection criteria. In addition to preview for the upper options, **Place by refdes** allows further options such as class filter and pin count.

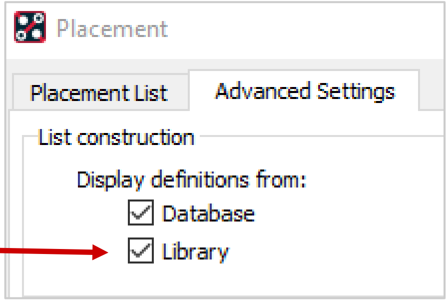
Placement Template

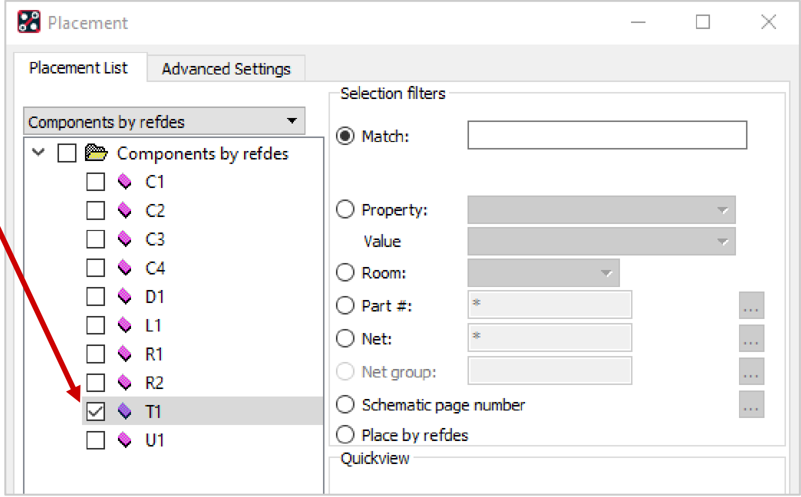


Lab: Place Manually (I)

In this chapter we will elaborate on the flow and options for component placement. We will use data generated in previous chapter **constraints.brd.** Placement picture on previous page is our template. After loading netlist (Import > Logic) information about components used in the design is available. With placement task the footprint will be loaded into board file.

1. Please open **constraints.brd**
2. **File > Script > Browse placement.scr > Replay**
   Optimization of graphics representation
3. **Place > Manually…**
4. Select library under **Advanced Settings**
5. Chose transformer **T1.**
6. **RMB > Rotate** (Pin 1 and 5 to the left)
   Alternative **Short Key R** as defined in Chapter 2.
7. **LMB** to position T1.
   Please place transformer in top left corner of board. Please note **P** in updated placement list right after placement.
8. Close placement window with **OK.**
9. Don't place any further components yet.





Filter Parameters

Before we complete placement, we would like to focus on different options of selection filter. Especially for larger boards multiple filter criteria are very efficient during manual placement.

Selection filter allows definition of placement options by limiting components which are later available for selection.

Please test above mentioned options with our test example **before** you have completed placement. Some options will not be available after a completed placement.

Choose option which fits best to your works tile and needs.

Match	Via REFDES and wildcard "*" followed by TAB key you can select for example capacitors only
Property	Defined properties and values including user properties
Room	Placement by room property for floor planning
Part #	Placement by part number
Net	Placement of parts which are connected to a specific net.
Schematic page number	Only available for Design Entry HDL
Place by refdes	Additional selection criteria like class property or pin count

Additional Commands (I)

During or after placement there are additional functions required to realize or modify a PCB design.

Below you can find most important functions accessible via **Edit > …** or by illustrated icons.

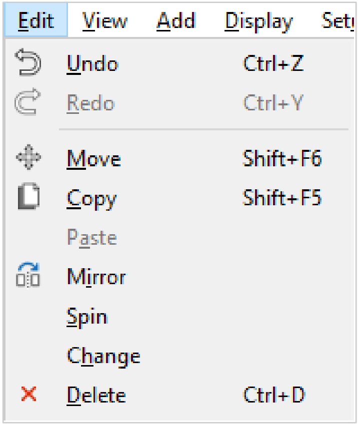
Move → Move of components

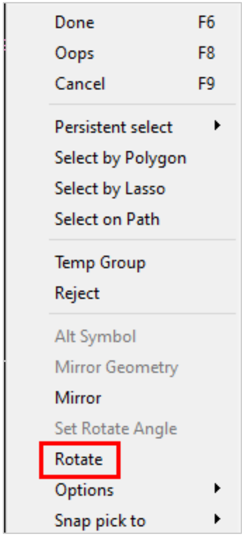
Copy → Copy of elements

Mirror Move parts from top layer to bottom layer

Spin Rotate elements

Delete → Delete elements





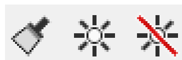
Tip
Please pay attention to **Find Filter** and extensive options in **Option Control Panel** for all functions.

Many functions are accessible via right mouse button. E. g. **Rotate** is a sub function of **Move** or **Place**.

Additional Commands (II)

Ratsnet OFF / ON 

Guidelines on / off (**Display > Show / Blank Rats >…)**

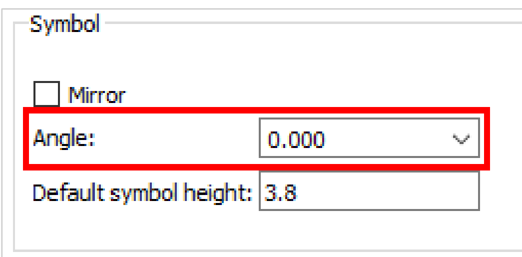
Assign Color 

Permanent color assignment of any elements

Highlight / Dehighlight

Usage of Find Filter and Option Panel
Also via **Display > Assign Color / Highlight / Dehighlight**

Setup > Design Parameters…



Mirror and **Angle** allow a presetting under
Place > Manually > Design related to placement
TOP / BOTTOM and rotation.

Complementary elements for placement:

- By using room property floorplanning could be already prepared in schematic
  ➔ **Setup > Outlines > Room Outline…**
- Package Keep outs / Keep ins / Height are restrictions for components
  ➔ **Setup > Areas > Package Keep out / Keep in / Height**

Quickplace

Start Quickplace via **Place > Quickplace…**

Quickplace is a very flexible and universal usable tool to make component placement task more effective.

With one click you can verify if all library elements for parts in netlist are available. To do so, use commands below:

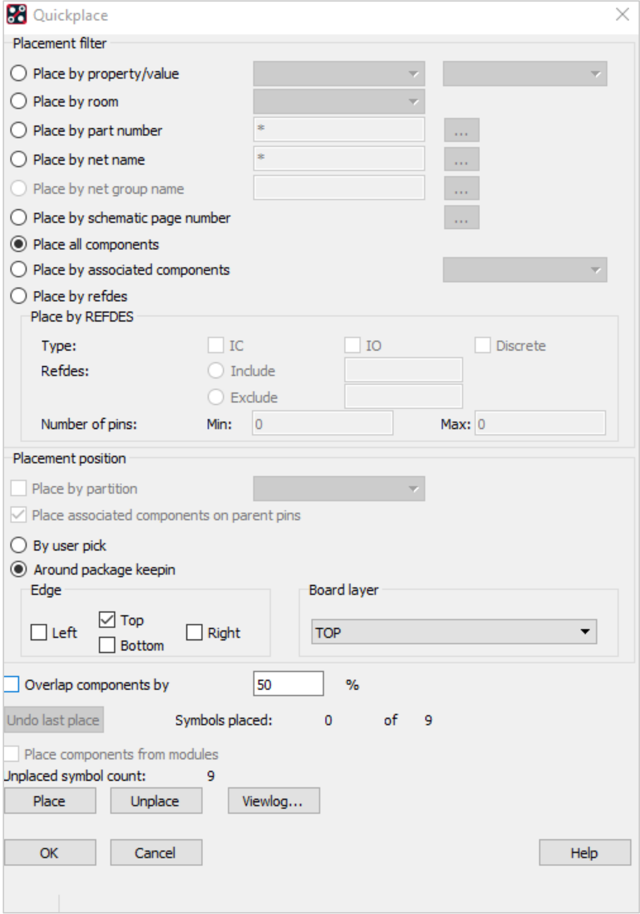
- Placement Filter set to **Place all components**
- Placement position to **Around package keepin**

and then: **Place**

Unplaced Symbol count gives a message if components could not be placed because of missing library elements or an incomplete library path.

Place Manually function is used to check which parts these are.

All other options are self-explanatory, although some of them require definition of corresponding properties in schematic.



Tip

Unplace is only possible as long **Quickplace** has not been confirmed via OK.

Lab: Place Manually (II)

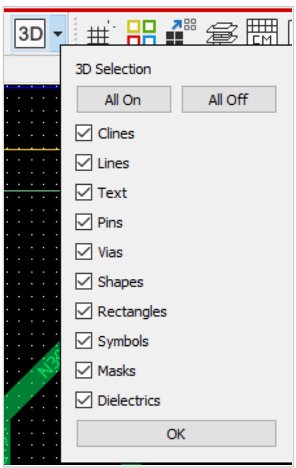
1. Choose **Place > Manually…**
2. Please note that T1 is not listed anymore.
3. In selection filter please select ROOM and chose Room Gleichrichter.
4. Parts C1, C2, D1 and L1 are listed in selection window.
5. Select **D1** and rotate by 180 degree.
6. Place via command line. Enter: **x 30 45** and **Enter.**
7. Select capacitors and place them with pin 1 to the left (270 degree) to position: 45 45 and 45 35.
8. Select L1 and place coil with pin 1 to the top (270 degree) to position: 37.5 40.
9. Now select **ROOM** in selection filter and chose **U-Regler** as room.
10. All remaining components are listed. Please select all.
11. Place one after another at: 45 25; 45 20; 45 15; 45 10 (90 degree); 37.5 12.5 (90 degree), **RMB > Done.**
12. Once again **Place > Manually.** List for **Comp by Refdes** is empty. All components are placed now.
13. Close placement window with **OK.**
14. **Edit > Change** and select in Option Panel **Text Block + 6**, set Find Filter to **Text only**.
15. Click on each REFDES or drag a rectangle over a group. All REFDES have now **same** size.
16. **Edit > Move** and set Find Filter to **Text only.**
17. Move all REFDES as illustrated and use rotate command (e. g. RMB).
18. Save result as **placed.brd**.

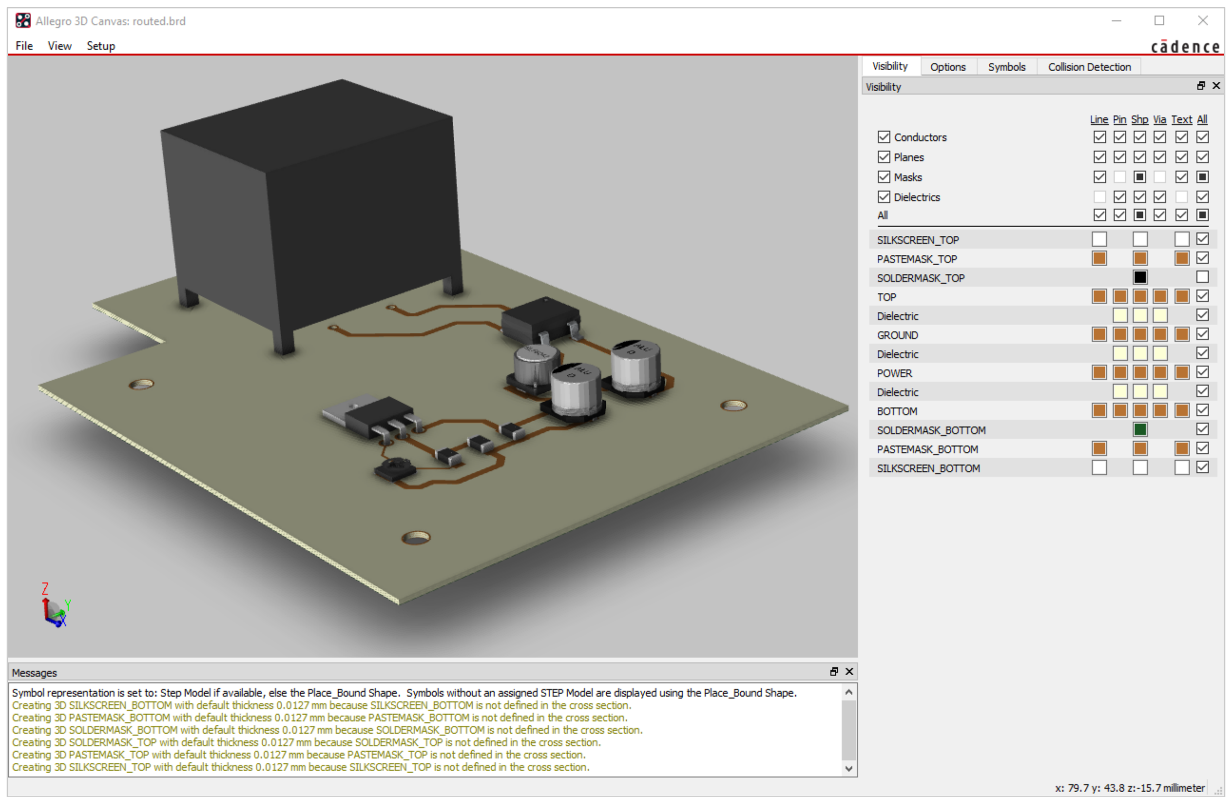
3D Canvas

3D Canvas

Via **View > 3D** or 3D you get access to 3D Canvas.

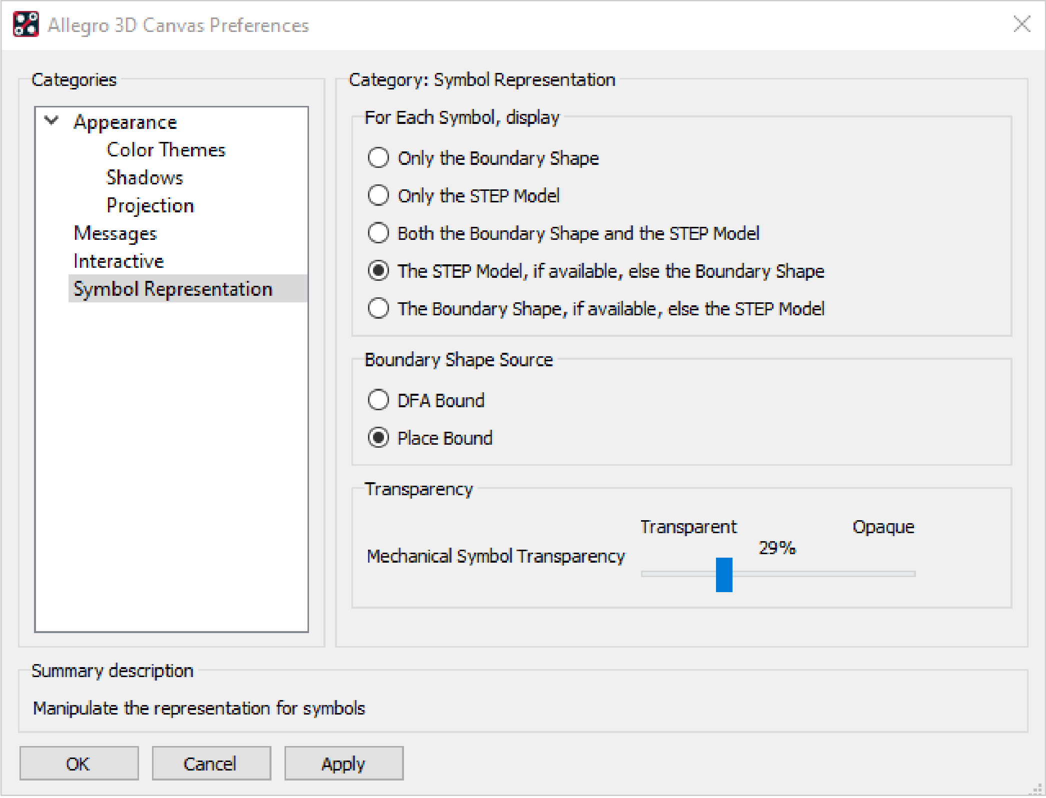
Use drop-down menu on 3D button to select what should be displayed in 3D Canvas.





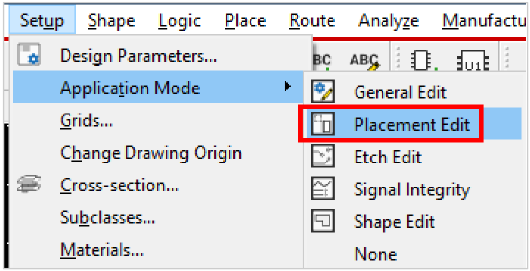
3D Canvas

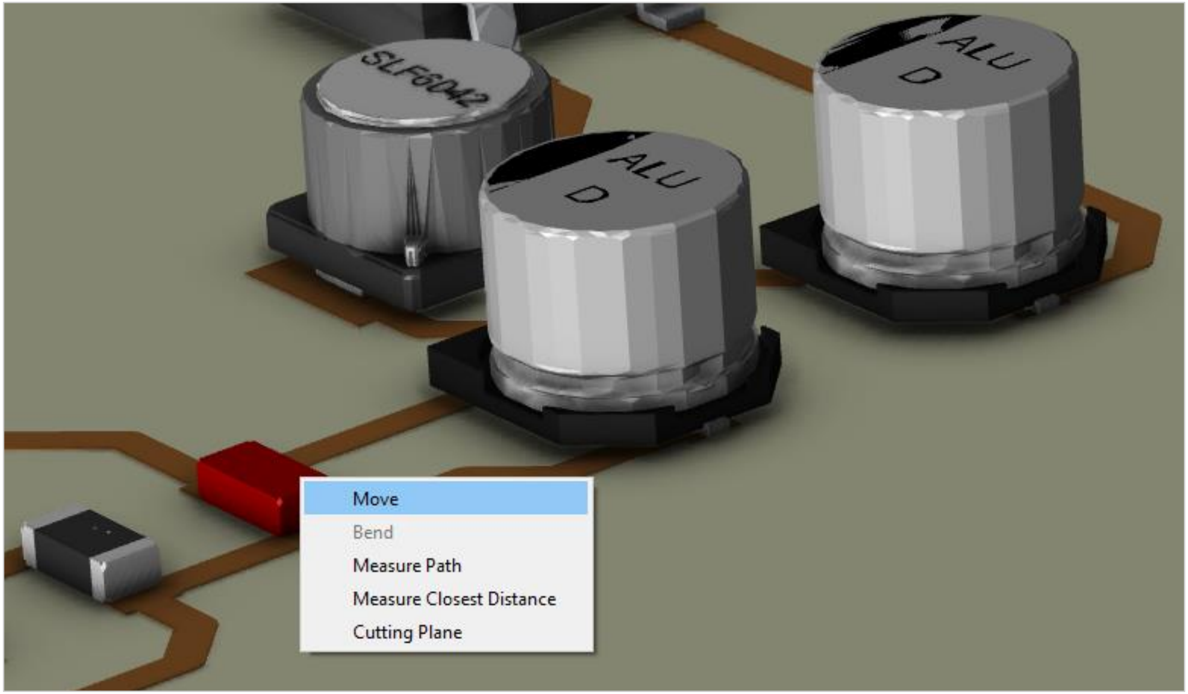
Define visual representation of parts under **Setup > Preferences > Symbol Representation**.



3D Canvas

When PCB Editor is in **Placement Edit** Application Mode, parts can be moved in 3D Canvas using **RMB > Move**.





FlowCAD

Routing

Routing

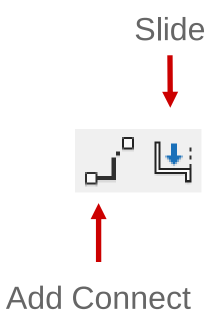
Routing is layout of copper traces based on netlist. It can be performed interactively or automatically (with appropriate license).

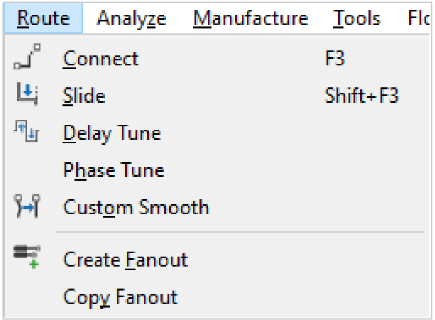
Both methods are available via icons or pull-down menus.

Add Connect (**Route > Connect**):	Manual routing of electric connections
Slide (**Route > Slide**):	Movement of existing traces
Create Fanout (**Route > Create Fanout**):	Upfront creation of Fanout / PinEscape
Custom Smooth (**Route > Custom Smooth**):	Trace optimization
Edit Vertex (**Edit > Vertex**):	Add / delete vertices of traces

Auto Route Param / Route (**Route > PCB Router > Route Automatic**):
Opens parameter form and start PCB routers in background.

In this tutorial we will focus on interactive routing.





Routing Grid

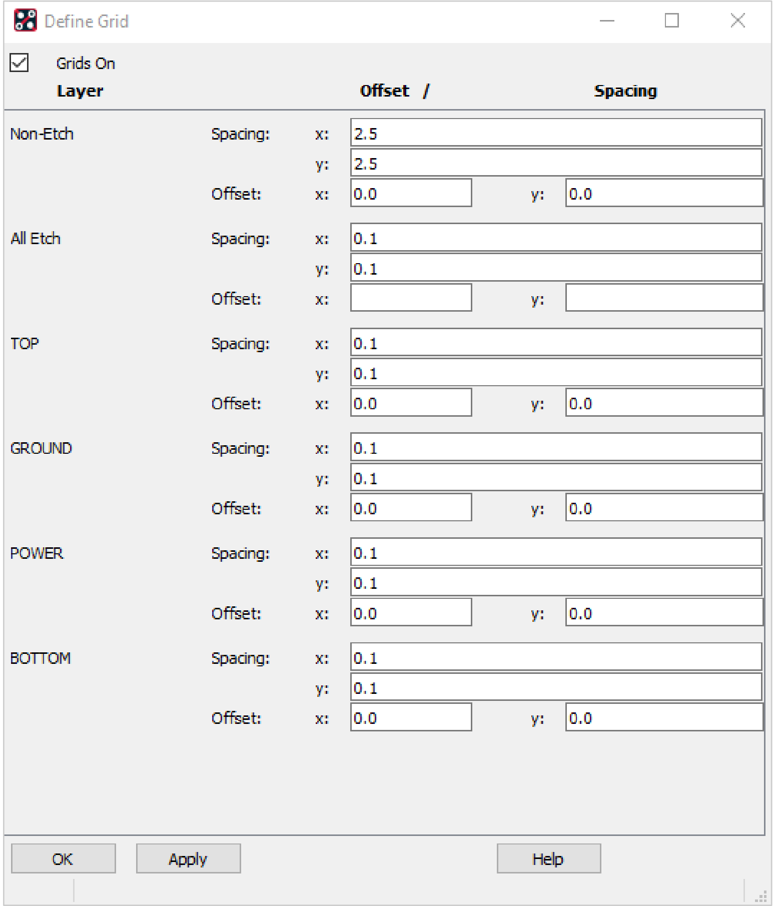
Etch grid will be immediately displayed if a route command like **Route > Connect** is executed. During interactive routing with the mouse it is used as snap grid.

Choose **Setup > Grids** and enter all values in section All Etch as displayed.

Grids On box does control visibility of grid.

Entries in All Etch apply to all layers.

If different grids on different layers are desired, you can enter grids for each layer.



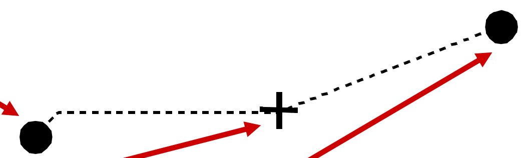
Attention

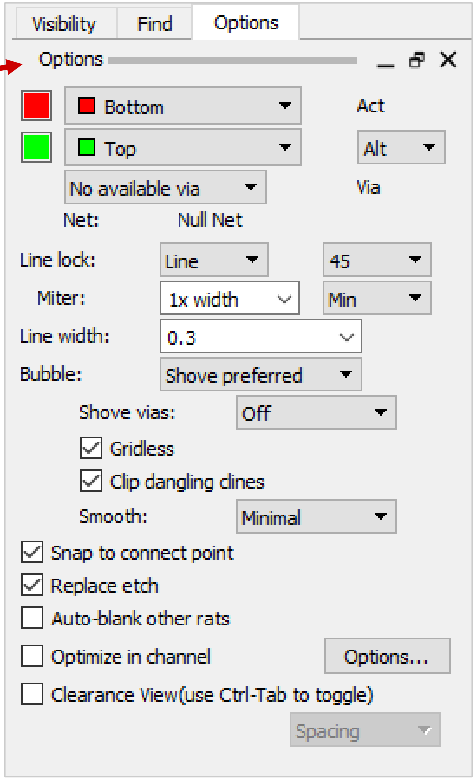
With different grids on different etch layers a via is only possible on a common multiplier grid.

Lab: Routing

Routing:

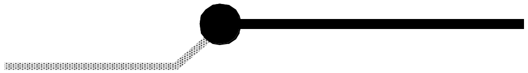
- Routing start via: **Route > Connect** or 
  – Routing behavior is defined on Options panel.
- Select start point with **LMB,** i. e. T1 Pin 9.





- **LMB** to fix planned path.
- **LMB** on endpoint to finalize trace.
- Another connection via **RMB > Next.**
- Finish connect command by **RMB > Done**.
- **RMB > Oops** undoes last step.
- **RMB > Cancel** will finish command without saving changes.

Adding Vias:

- **RMB > Add Via** or double click 
- Setting of used via and target layer will be defined in Options panel.

Routing Options

In chapter design rules we already discussed some pre-settings like trace with or trace clearances. When we start routing traces and online DRC is active, we already get active support from the tool. Predefined trace will be set correctly as a parameter. We can change trace within defined tolerance. If we are outside of this tolerance will get an immediate DRC warning from the system and can respond appropriately.

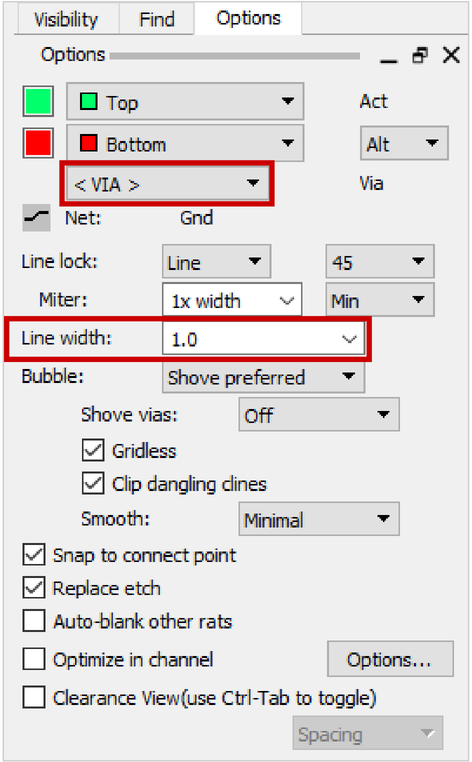
If command **Route > Connect** was started without a selected pin or guideline, Options panel will show default values:

Via: No available Via (because there is no net selected)

Net: Null Net (because there is no net selected)

Line width: 0.3 (default value, if there is no net selected)

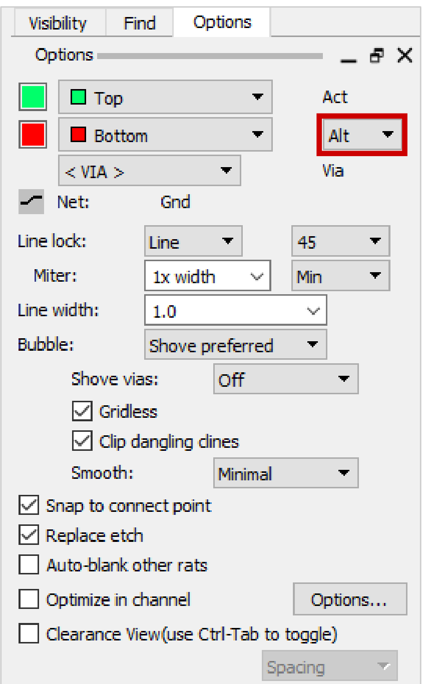
As soon as a net is selected (click on pin), valid values for this net will be displayed.



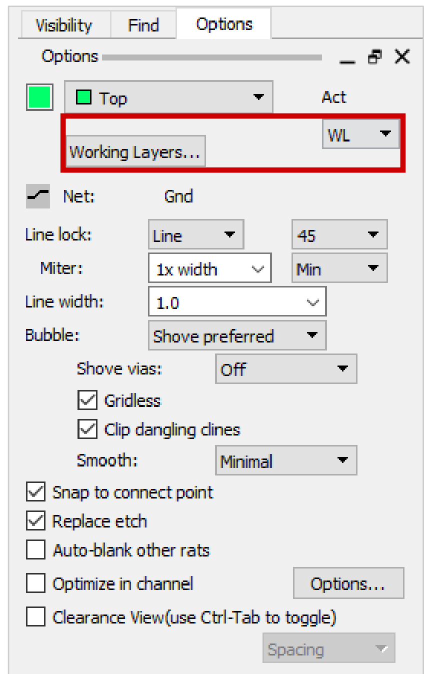
Alternate Mode / Working Layer Mode

In **Alternate Mode** we select two layers in Options panel. During routing it is possible to switch between them.

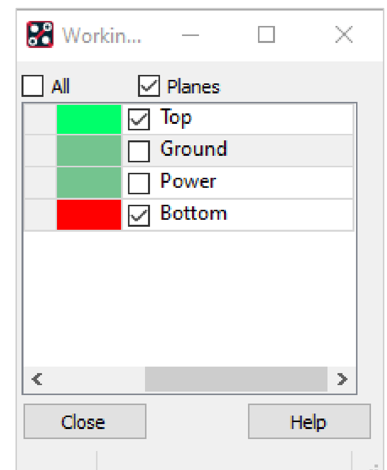
Working Layer Mode is specific for working interactively on high layer count multilayer boards. If this mode is activated, you can define any layers preferred used during interactive routing. If multiple layers are enabled a form will appear while setting a via to choose desired target layer. Plane layers can be completely excluded and disabled.



Alternate Mode



Working Layer Mode



Net and Line Options

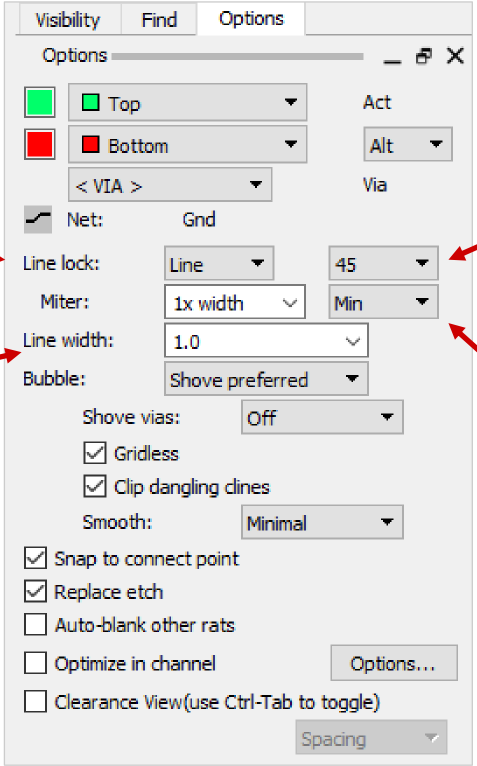
Line lock:

Line: Routing of straight lines

Arc: Routing of arcs

Line width:

If there are specific values via property **Min_line_width** or **CSets** to selected net assigned, they will be used. Otherwise default values from Constraint Setup will be used.



Angle:

45 / 90: Fixed angle

Off: Any angle

Chamfer corners:

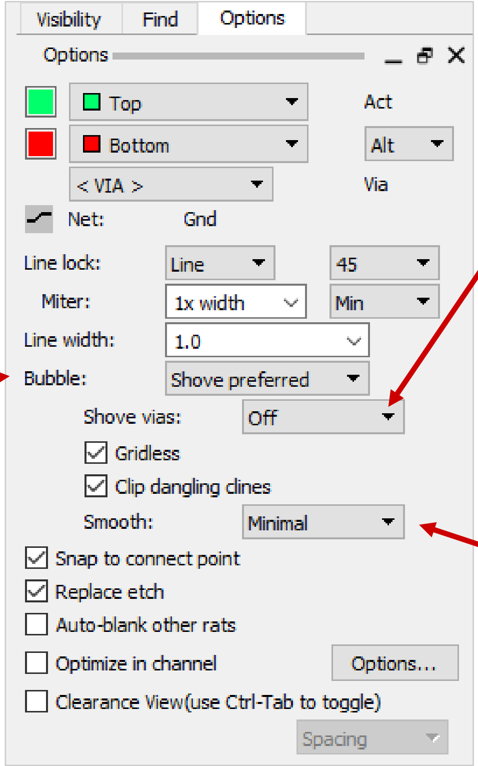
Min = no length limitation

Fixed = left value will be used (1 x width)

Tip

Please note that settings of parameter Line Lock and Miter influence each other. Also arcs can be impacted by Miter settings.

Push, Smooth



Bubble:

Setting defines behavior with existing traces.

Off: It will route over existing traces.

Hug Only: Line will align next to an existing trace.

Hug preferred: Line will prefer to align with existing traces.

Shove preferred: Other traces will be pushed.

Shove vias:

Off: Vias will not be moved.

Minimal: Vias will moved just a bit.

Full: Vias will be moved.

Smooth:

Off: Existing traces will not get touched and smoothened.

Minimal: Minimum smoothening of existing traces.

Full: Better smoothening

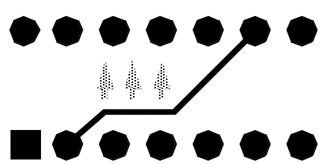
Super: Maximum smoothening

Editing of Existing Traces (I)

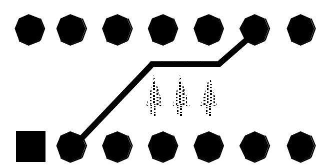
There are extensive possibilities to modify existing traces in PCB Editor.

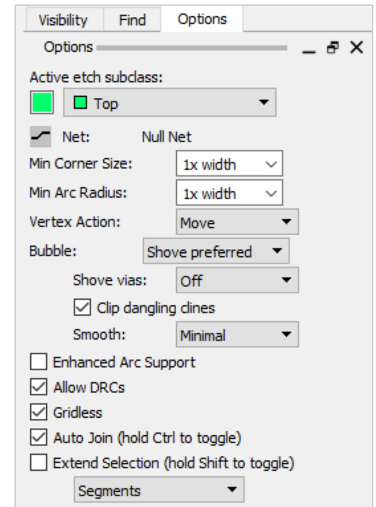
Route > Slide: Movement of traces adjacent to chosen settings

Before



After

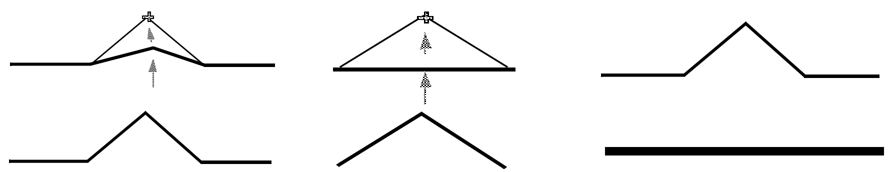




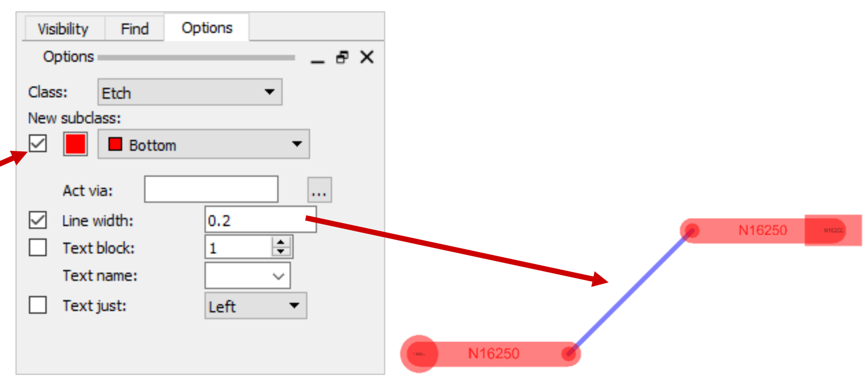
Edit > Vertex: New vertices or movement of existing vertices

Before

After



Edit > Change: Changes of layer or width of existing trace segments. For layer changes necessary vias will be generated automatically.



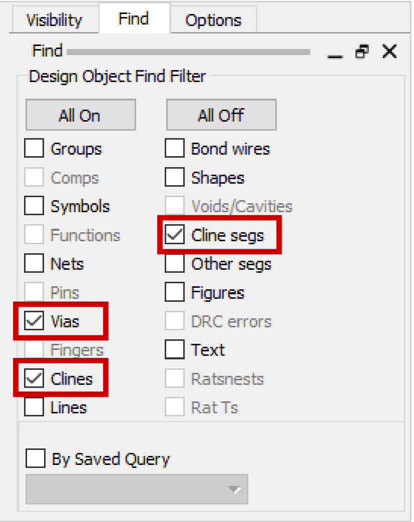
Editing of Existing Traces (II)

Delete: Deletes traces and vias

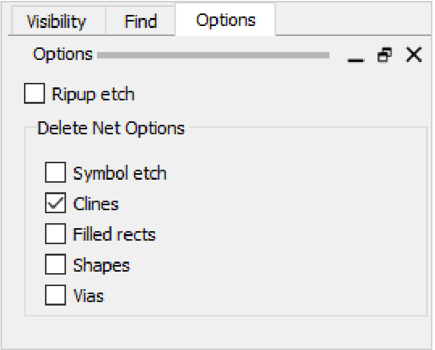
Cline Segs:
Deletes segments of a net

Clines:
Deletes all segments of a net with exception of vias

Vias:
Deletes vias



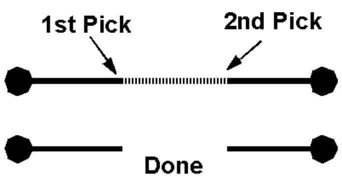
Ripup Etch:



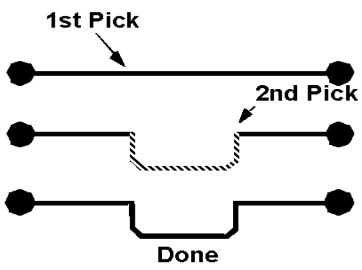
Use **Ripup etch** and **Clines** to delete all segments and vias between pins across multiple layers.

Cut option (**RMB > Cut**): Allows to select dedicated sections of a segment

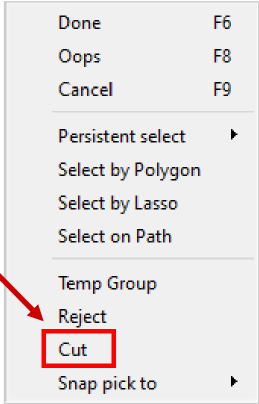
Edit > Delete



Route > Slide



Edit > Change (width)

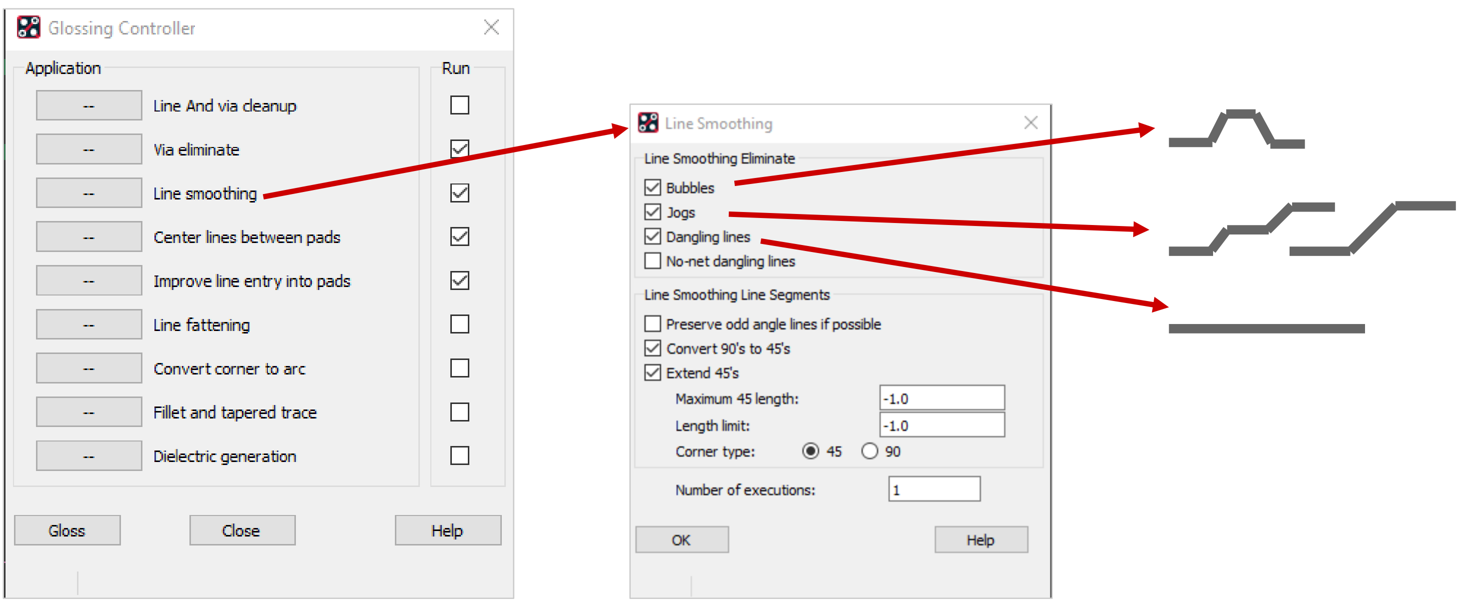


Glossing

Glossing is improving trace structure for manufacturing. Glossing can remove redundant bubbles, jogs and dangling lines automatically.

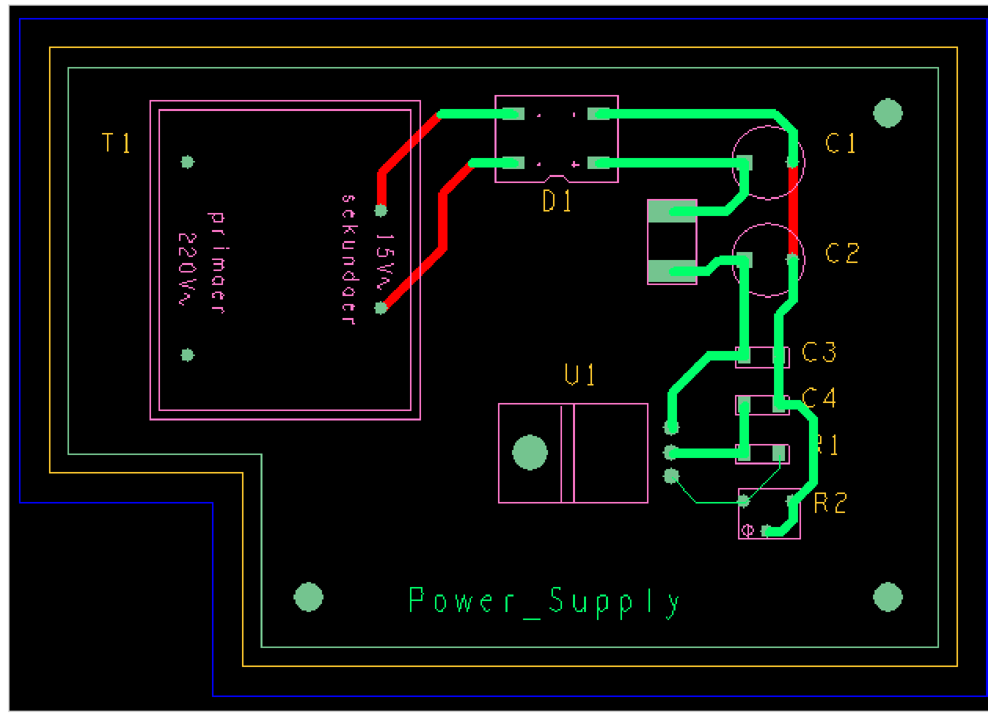
In addition OrCAD provides much more glossing capabilities which are not mentioned in this tutorial.

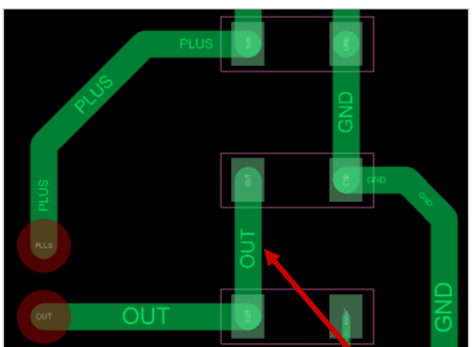
Route > Gloss > Line smoothing…



Lab: Routing

1. Please load **placed.brd** from previous exercise and use learned skills to route the board according to template.
   Green is Top
   Red is Bottom
2. Place text **Power_Supply** on top layer.
3. Save the routed board under **routed.brd**.





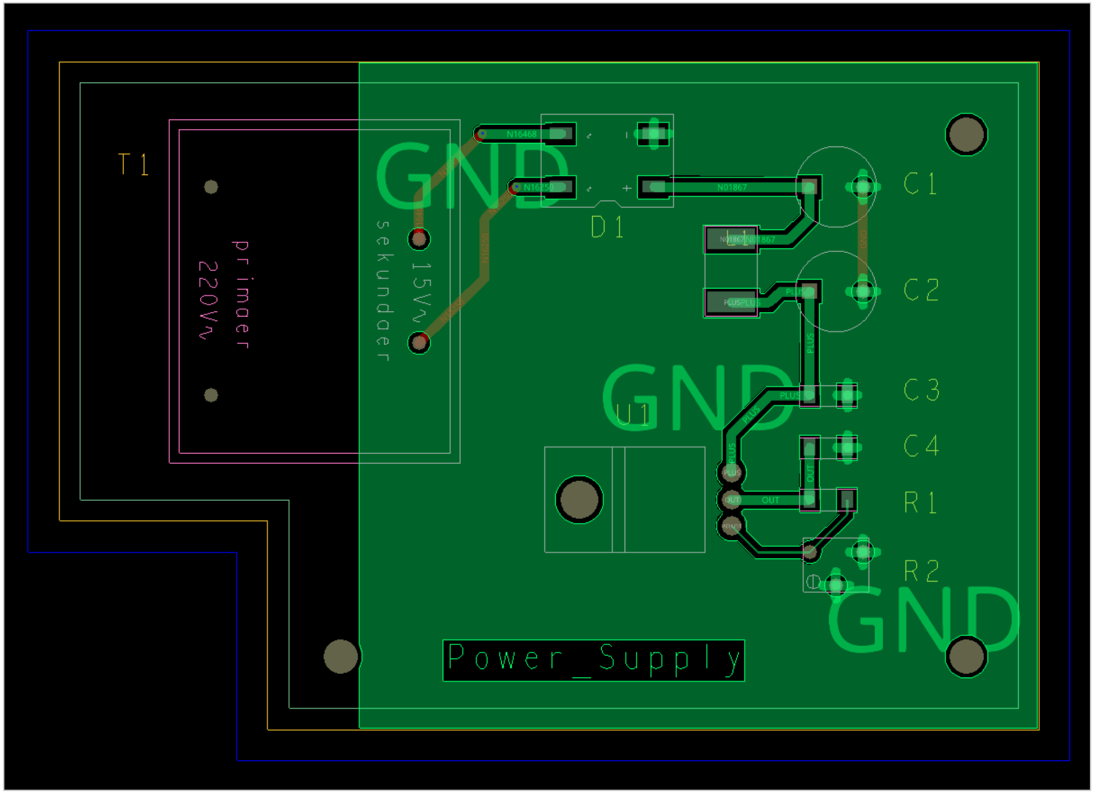
Net names can be displayed under:
Setup > Design Parameters > Display > Display Net Names.

Copper Areas

Copper Areas (Shapes)

Copper areas, in OrCAD named shapes, play a significant role for power distribution and shielding.

PCB Editor can handle both static and dynamic shapes. Dynamic shapes are recalculated in real time by any modification. This could be the shape itself, component movement or additional component or routing a trace through the shape.

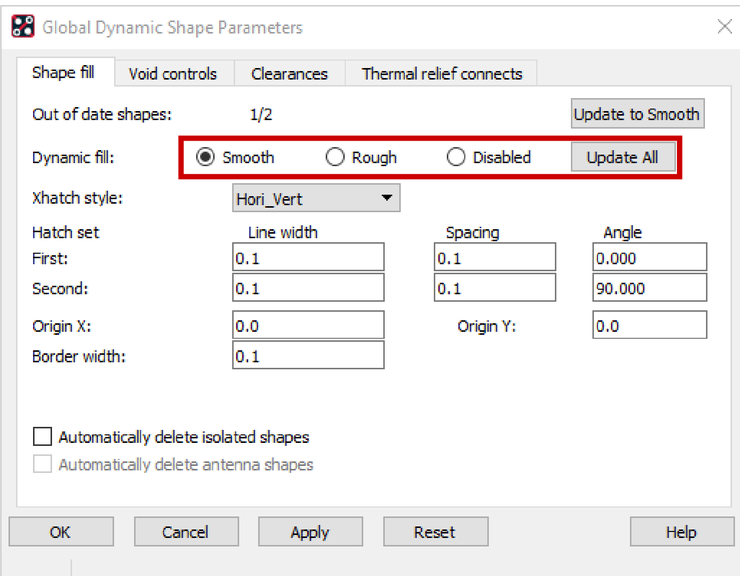


Tip

Usage of dynamic shapes is much more comfortable and safer.
PCB Editor takes all clearance rules in consideration and cuts out elements dynamically.

Global Dynamic Parameters: Shape Fill

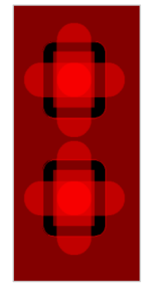
Under **Setup > Global Dynamic Params…** you can define ordinary parameters for all new shapes.
If necessary, you can adjust parameters for each individual shape.



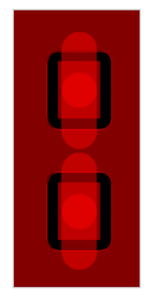
Tip

Grid pattern shapes like used for flex boards can be defined with **Xhatch** style.

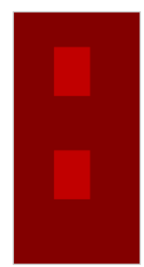
Dynamic fill:



Smooth corresponds gerber output.



Rough calculates shapes with less precision. Saving time when handling large designs.



Disabled: Shapes don't get recalculated. Via **Update to Smooth** all shapes get recalculated.

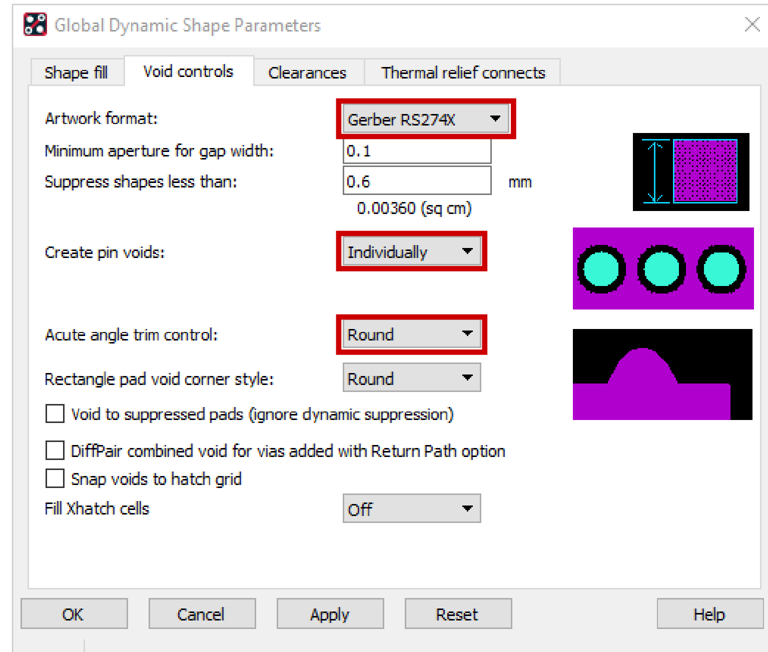
Global Dynamic Parameters: Void Controls

Settings in this section impact implementation of shapes.

Artwork format should be set to **RS274X**, this is industry standard. We will talk about artwork format related to manufacturing outputs in this chapter later.

Create pin voids:
Individual: Pins get individual cutouts.
In-line: cutout will be merged for more pins.

Acute angle trim control:
Does impact handling of corners depending on angle.

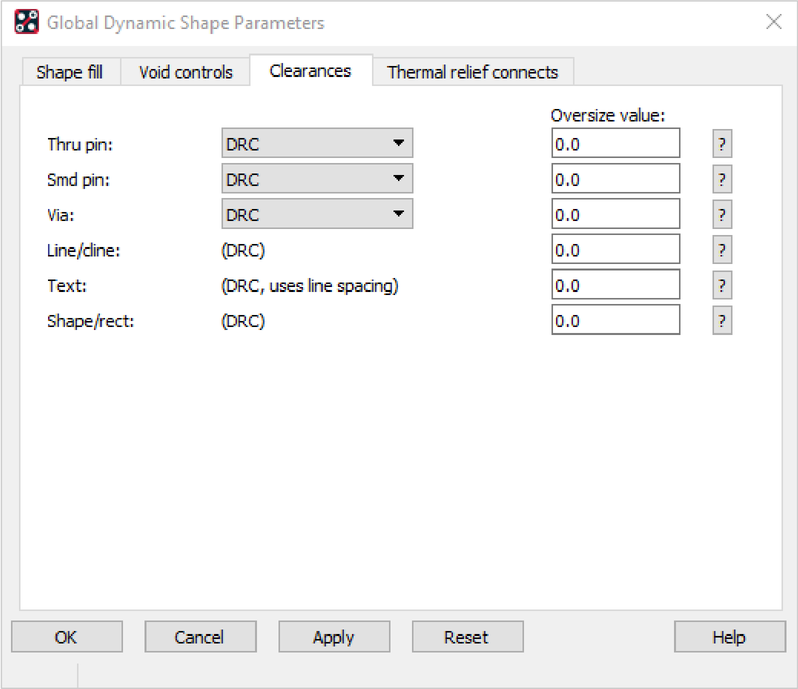


Tip

Generally, all shapes of a board have same format assigned. Therefore this bullet is excluded from individual parameters of shapes.

Global Dynamic Parameters: Clearances

Clearances define clearance (cut out) for each pad type. Default is DRC, meaning values from Constraint Manager are used.

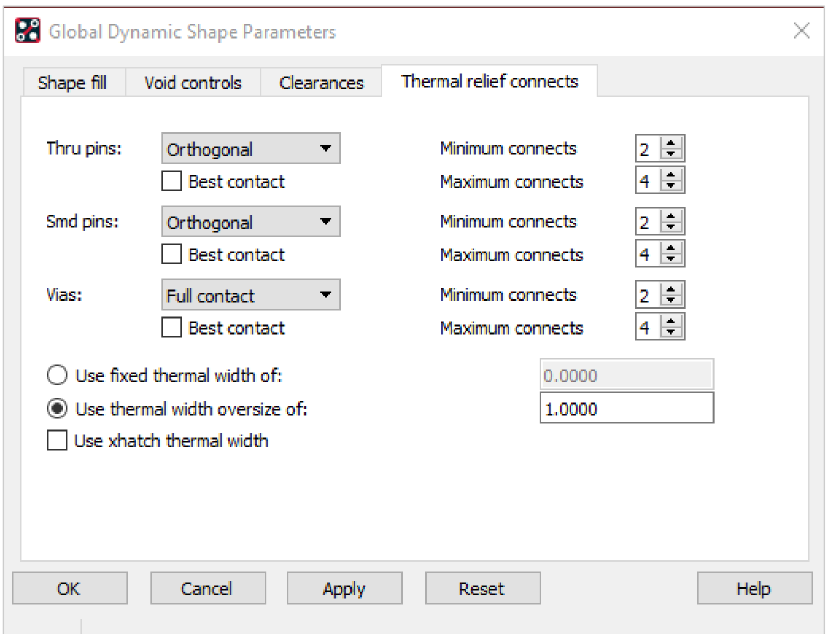


Tip

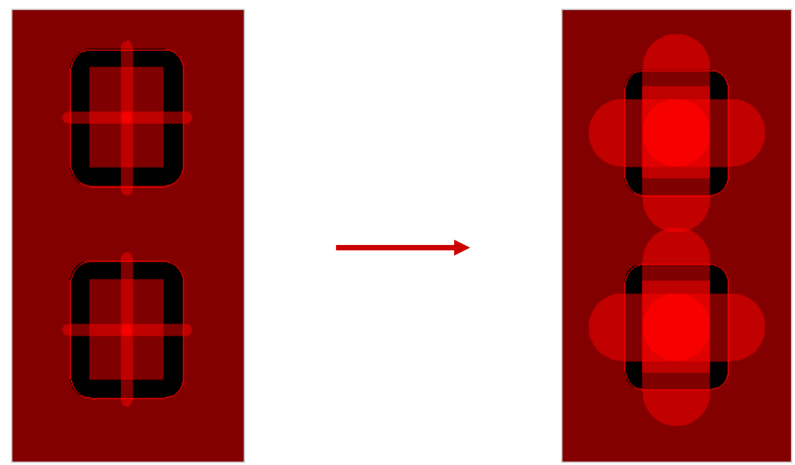
Please avoid oversized values.
Correct clearance values should be predefined in Constraint Manager.

Global Dyn. Param.: Thermal Relief Connects

Here you define thermal reliefs. There are different definitions for **Thru Pins, Smd Pins and Vias.**



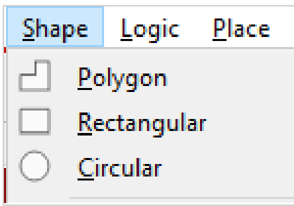
This value allows expansion of thermal reliefs compared to original design rules.



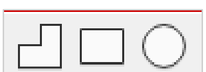
Shapes filled as Xhatch can use Xhatch line width also for thermal relief width.

Adding Shapes

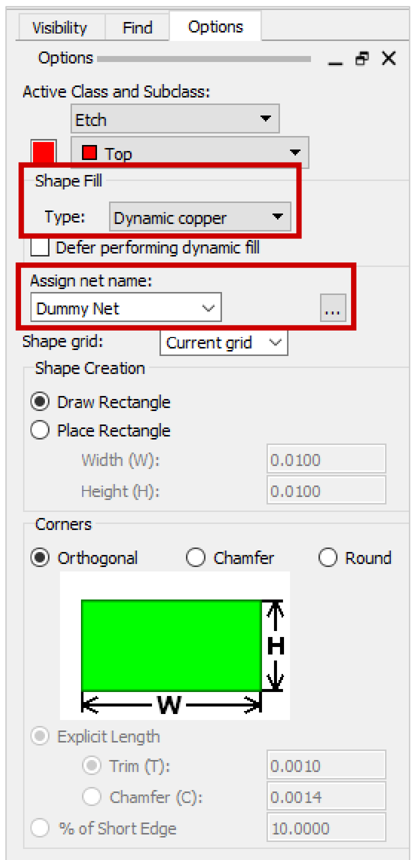
You can add shapes via **Shape** menu.

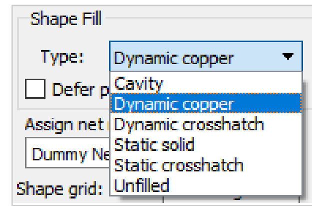


Shape Buttons:

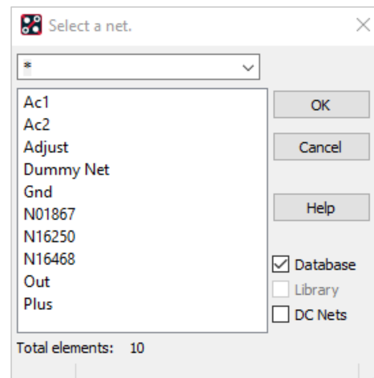


Shape Options:





You can choose between **Dynamic** and **Static** Shapes.



After outline definition **Dynamic Shapes** calculate shape automatically and keep clearances to all elements according to parameters. This task will be performed for every change too. For **Static Shape** recalculation needs to be triggered by the user.

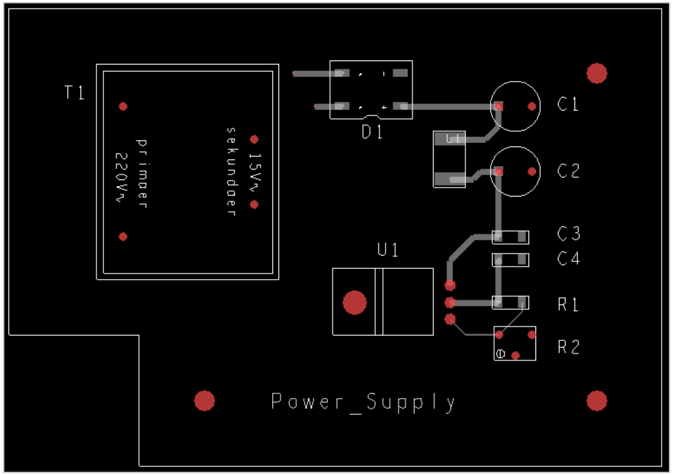
Tip

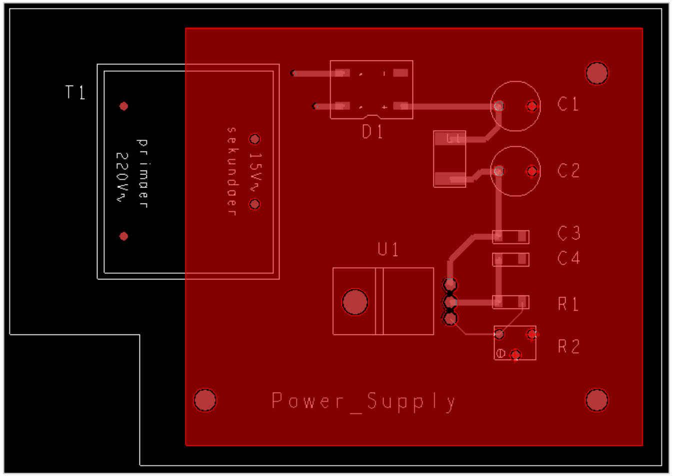
Please check points below before you generate shapes:

- **Global Dynamic Parameter**
- **Active Layer**
- **Chosen Net Name**

Lab: Shape Generation

1. Load board **routed.brd.**
2. Set gerber format under **Shape > Global Dynamics Parameters… > Void controls auf RS274X.**
3. Choose **Shape > Rectangle.**
4. Make ground layer visible **(Visibility).**
5. Set **Class** and **Subclass** to **Etch / GND.**
6. Set **Fill Type** to **Dynamic.**
7. Assign net name **GND** to the shape. Please use browser ... in option panel for this task.
8. Click **RMB** to define local parameters for the shape. This is only for positive shapes meaningful.
9. Drag a rectangle (2 x click with LMB) over right section of the board like illustrated.
10. Finish with **RMB** and Done.
11. Please note that shape outline is constraint by **Route Keep in** (yellow line). This results in a modified new shape outline.
12. Please repeat same procedure for Top layer.
13. Choose **Edit > Move** with filter on shape and move the shape. Please note that original shape contour is always in background preserved.



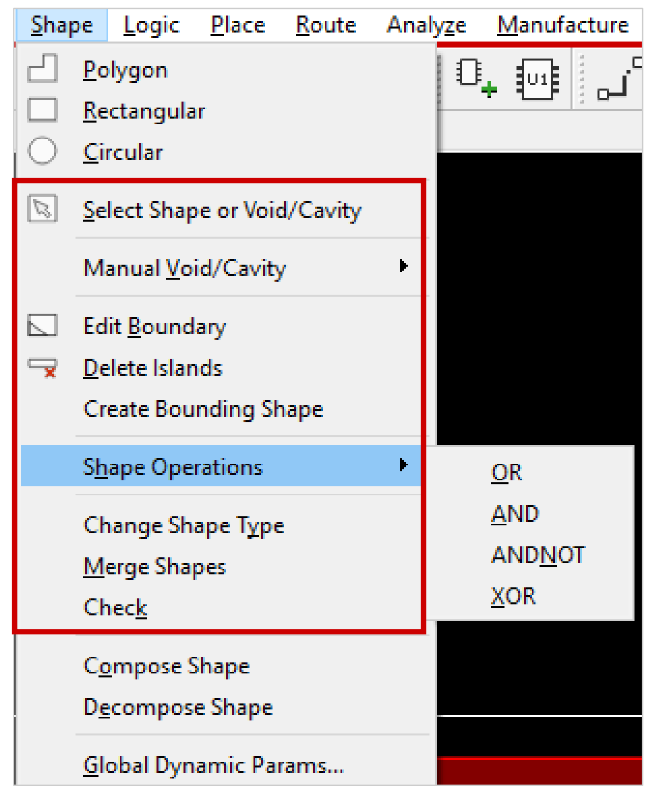


Shape Editing

In **Shape** menu many options for shape editing are available.

Tip

Please pay attention that you have set correct layer in options window for shape editing.
If necessary, you can assign a different net to a shape at any time.



Design Rule Check and Reports

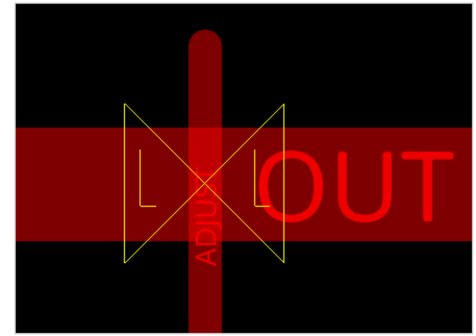
Design Rule Check

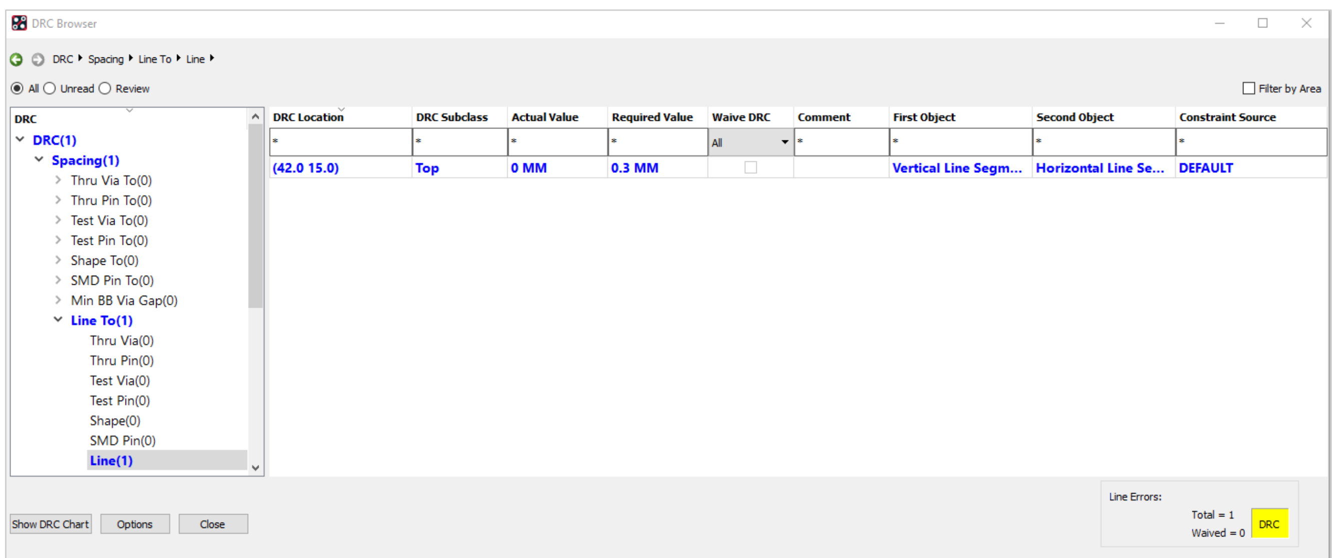
Violations of design rules get checked and marked online. Violations are marked with yellow DRC symbols (Butterfly shape) like illustrated on the right picture.

More detailed DRC information is available via **Display > Element** (set find filter to **DRC errors**).

All DRC violations are listed in DRC browser.
To open use **Tools > DRC Browser**.

By double clicking on coordinates the location of DRC will be centered in work window.





Display Status

Pop-up menu Display **Status** gives you an overview of design status.

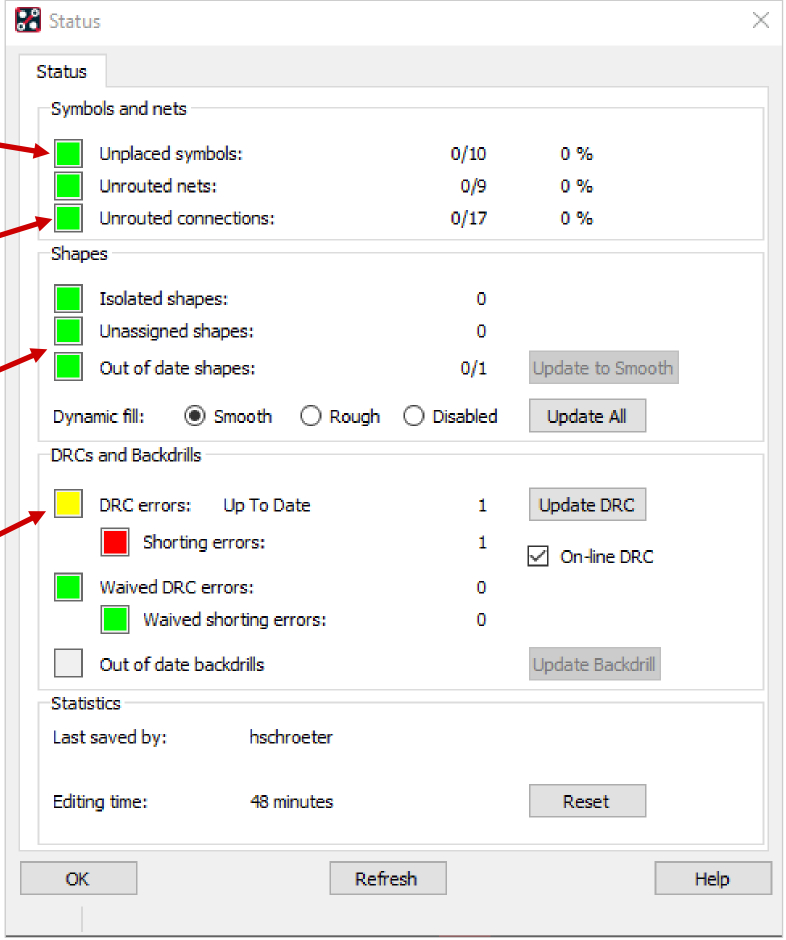
A click on colored buttons gives you more details. Some options allow a cross selection or high light by clicking on the buttons.

Are all components placed?

Are all nets routed?

Are all shapes error free?

Is DRC up to date and how many errors do exist?



Reports

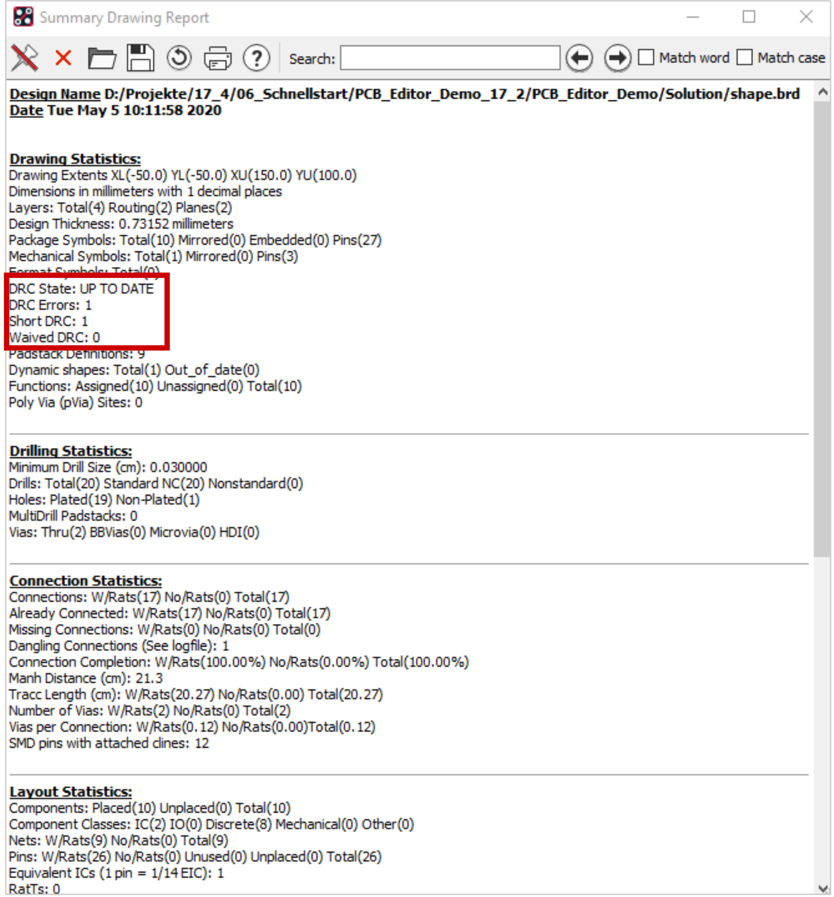
Another possibility to identify missing connections or design rule violations are reports.

Summary Drawing Report includes DRC status and possible incomplete connections.

Tools > Quick Reports > **Summary Drawing Report**

If you need more details regarding missing connections use **Unconnected Pins Report.**

Open connections are listed as links.
A simple click navigates you directly to sections in the layout.



Manufacturing Outputs

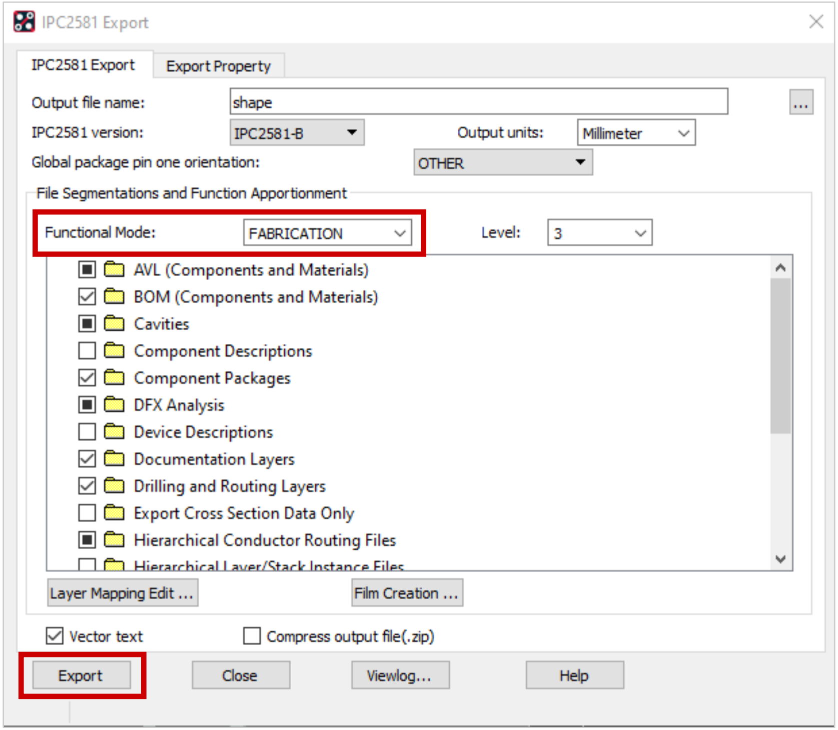
PCB Manufacturing Data – IPC-2581

In addition to classic output data like gerber and drill data, PCB Editor is able to export IPC-2581 and ODB++.

IPC-2581 as well as ODB++ container includes all necessary data for PCB bareboard manufacturing and PCB assembly.

Generate IPC-2581 via **File > Export > IPC 2581.**

Functional Mode allows to choose whether data should be written for PCB bareboard, assembly or other manufacturing steps.



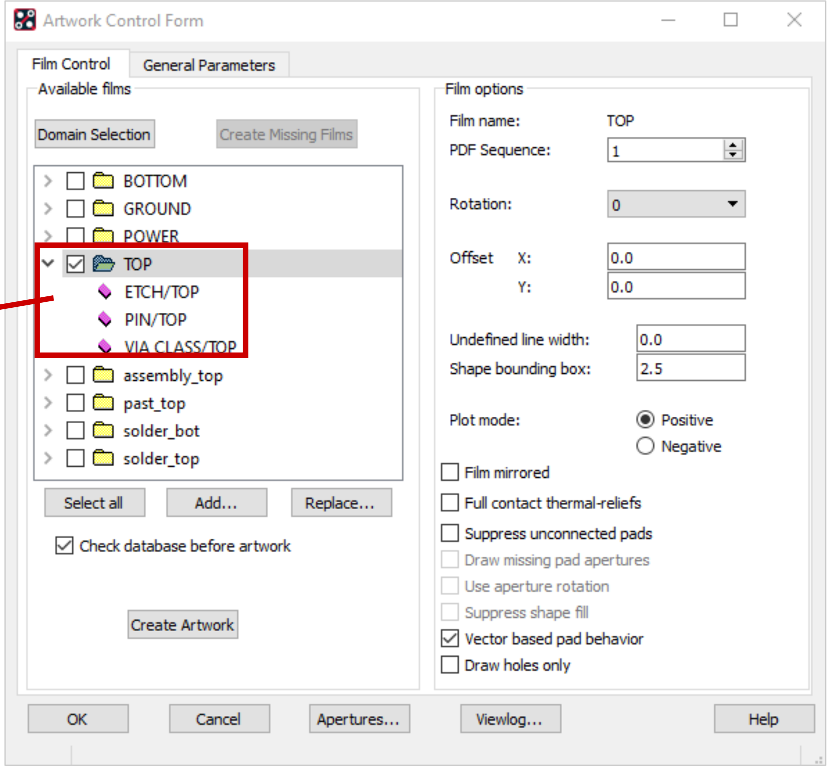
Manufacturing Output – Gerber Data

Classic output for manufacturing data consisting of gerber (artwork) and drill data (NC drill) is divided into two parts.

For gerber output of different gerber layers the necessary data will be assembled from PCB design database (xxx.brd).

For example gerber data for layer Top is assembled from design data Etch / Top, Pin / Top und Via Class / Top.

Top.art



Lab: Gerber Data (I)

Manufacture > Artwork opens user interface for gerber data output. In tab **General Parameter** base settings for output are defined.

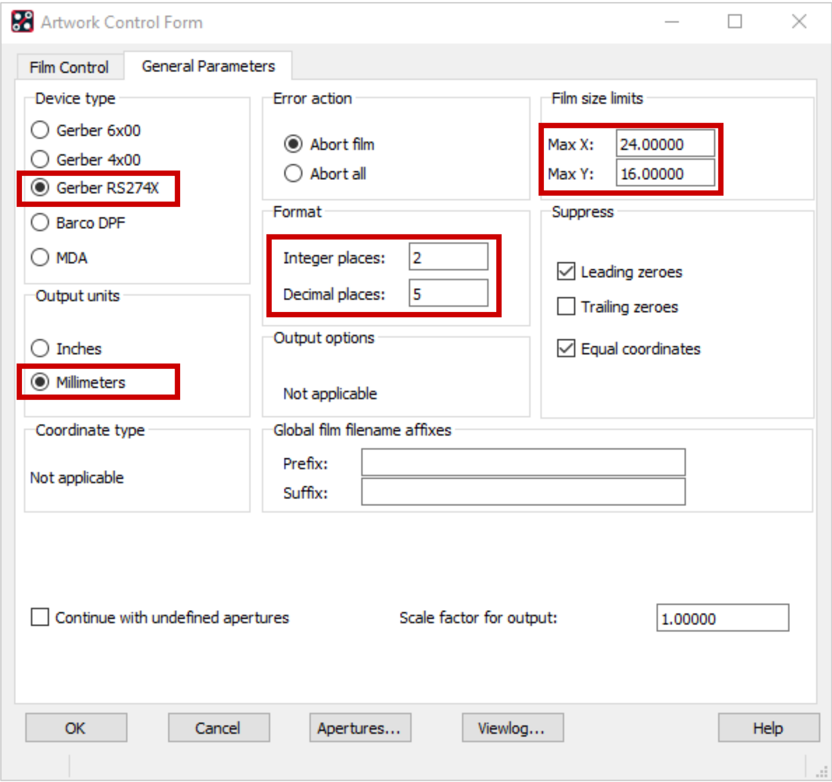
Please use following settings:

Devicetype (photo plot model):
Gerber RS274X – extended gerber

Film size limits (available plot size):
Film size: 24 x 16 mm

Format (Integer / Dezimal places):
Decimal places: 2,5

Output Units (units for the output):
Millimeters



Tip

Please use Extended Gerber (RS274X), this format contains apertures.

Use same units in board and outputs.

Do not use too low resolution (decimal places).

Artwork Film Options

Chosen options will be stored for each film record.

Filename: Name of film

PDF Sequence: Number of film in the pdf output.

Rotation: Film rotation (typically 0 degree)**.**

Offset: x / y offset

Undefined line width: Photo plot width for lines without dimensions. Always choose a value to avoid that structures will not be part of output data.

Shape bounding box: For negative planes, copper area will be generated with a negative offset related to entered value.

Plotmode: Negative only for negative planes used.

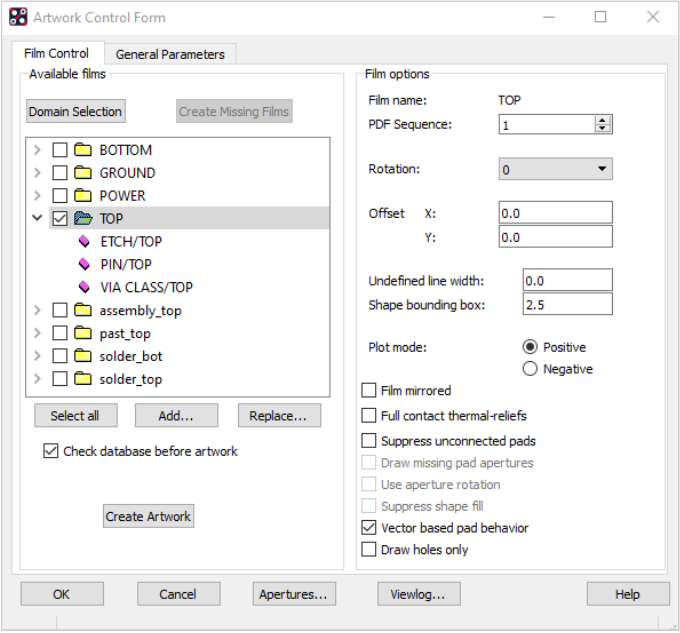
Film mirrored: Not standard (depends on bareboard manufacturer).

Full contact...: Suppress thermal reliefs on copper planes.

Suppress…: Suppress unconnected pads.

Vector based…: Laser plot behavior like vector plotter flashing for pads.

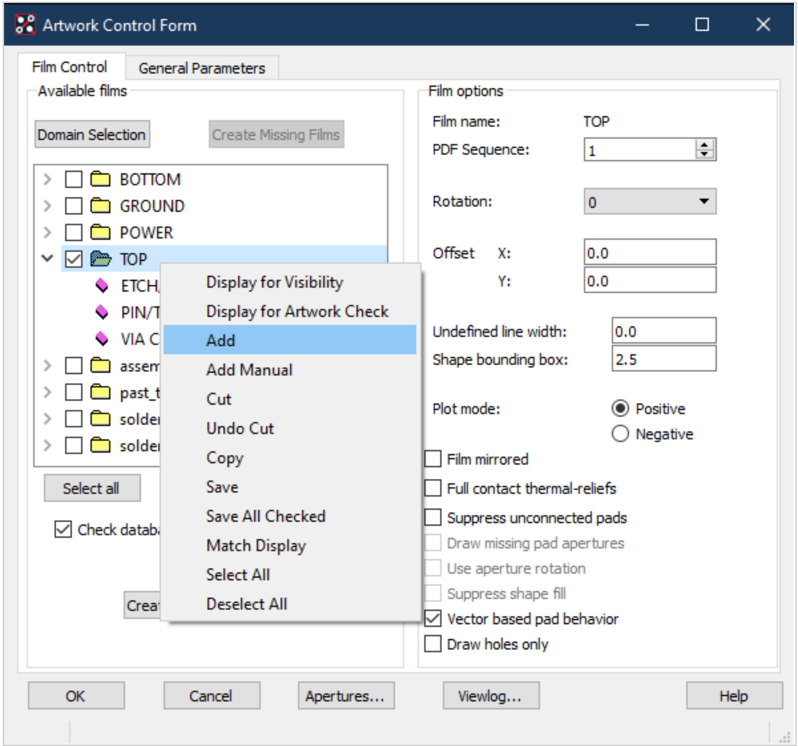
Draw holes only: Generation of shapes for drill holes. Applies only if only pins and / or vias on film are assigned.



Lab: Gerber Data (II)

In **Film Control Panel** all film records required for output must be defined. By default, all film records defined under **Setup > Subclasses… > Etch** appear here. Additional filmrecords like soldermask_top, assembly top etc. must be defined additionally.

- Set desired layers visible and all other layers hidden under **Display > Color Visibility** to define a new film.
- When you choose **RMB > Add** in Artwork Control Form, all visible layers will be combined to one film.



Lab: Drill Data

Menu to generate drill files:
Manufacture > NC > NC Drill

Auto tool select: Requires an extra file named **nc_tools.txt** for automated tool change of drilling machine.

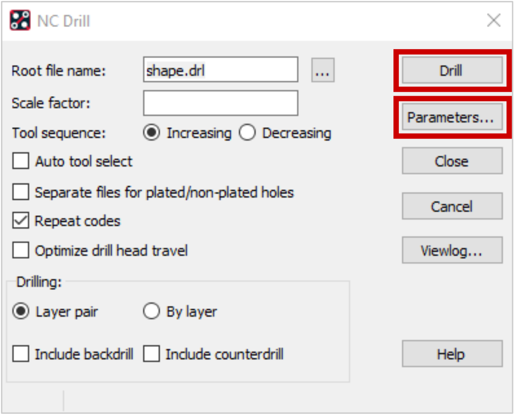
Separate files for...: Separated files for plated and non plated drill holes.

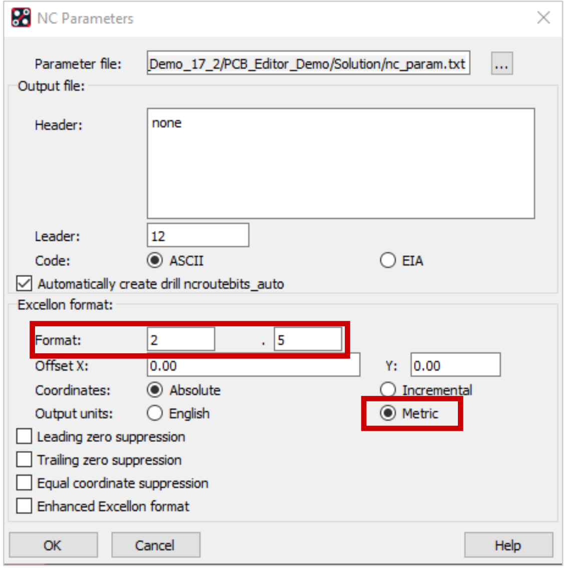
Repeat Codes: Should stay switched on.

Optimize drill…: Optimization of tool travel path.
Format and output units should be similar like settings for gerber files.

You can find more settings under **Parameters.**

Drill will start generation of drill data.



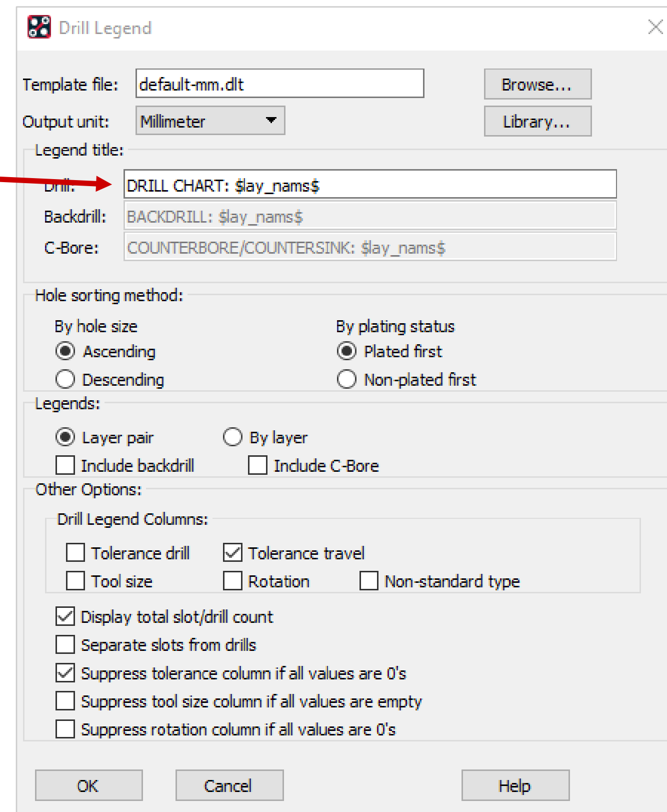


Documentation: NC Drill – Table

- Adding a drill table to layout use **Manufacture > NC > Drill Legend**.
- Via **Library** you can choose different templates files for different units.
- For blind and buried vias system automatically generates multiple drill charts and drill files.

Title of table and layer combination

DRILL CHART: TOP to BOTTOM			
ALL UNITS ARE IN MILS			
FIGURE	SIZE	PLATED	QTY
•	13.0	PLATED	506
◇	31.0	PLATED	12
+	38.0	PLATED	122
△	75.0	PLATED	8
□	110.0	NON-PLATED	5



Tip

If find filter is set to group, legend can be moved like one object.

Board Templates

Board Templates

This chapter gives a rough overview of how to create a mechanical board symbol or a board. Board templates are useful if there are identical structures in the layout process (outline, technology, or preplaced components).

A **mechanical board symbol** typically contains following elements:

- Boardoutline, keepin / keepout (route and package) via keepout, dimensioning
- Mounting holes

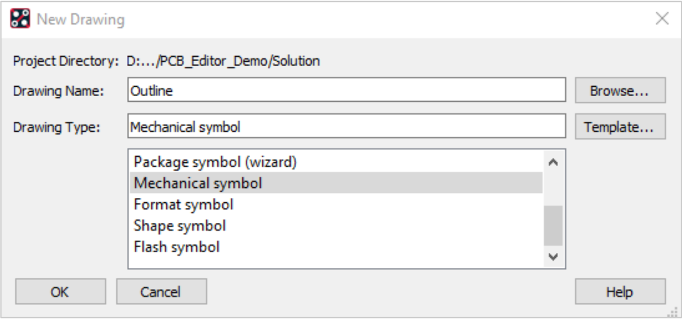
A **Masterboard** can contain the following elements:

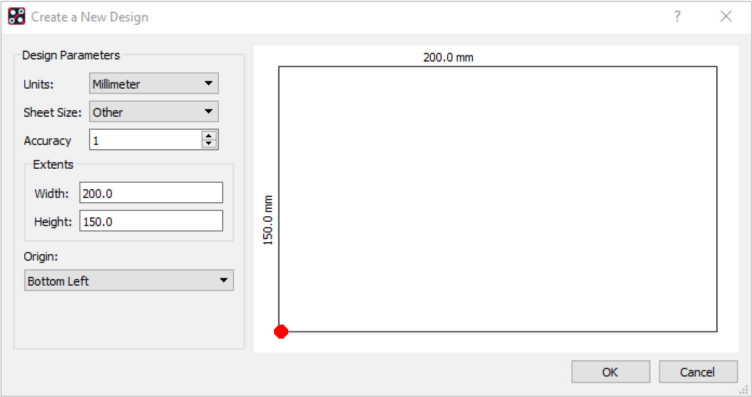
- Board symbol (.bsm)
- Drawing frame (.osm)
- Preplaced components, i.e. connectors (.psm)
- Technology constraints (clearance rules, trace with, etc.)
- Layer stack

Lab: Board Symbol

Follow steps below to generate a board symbol:

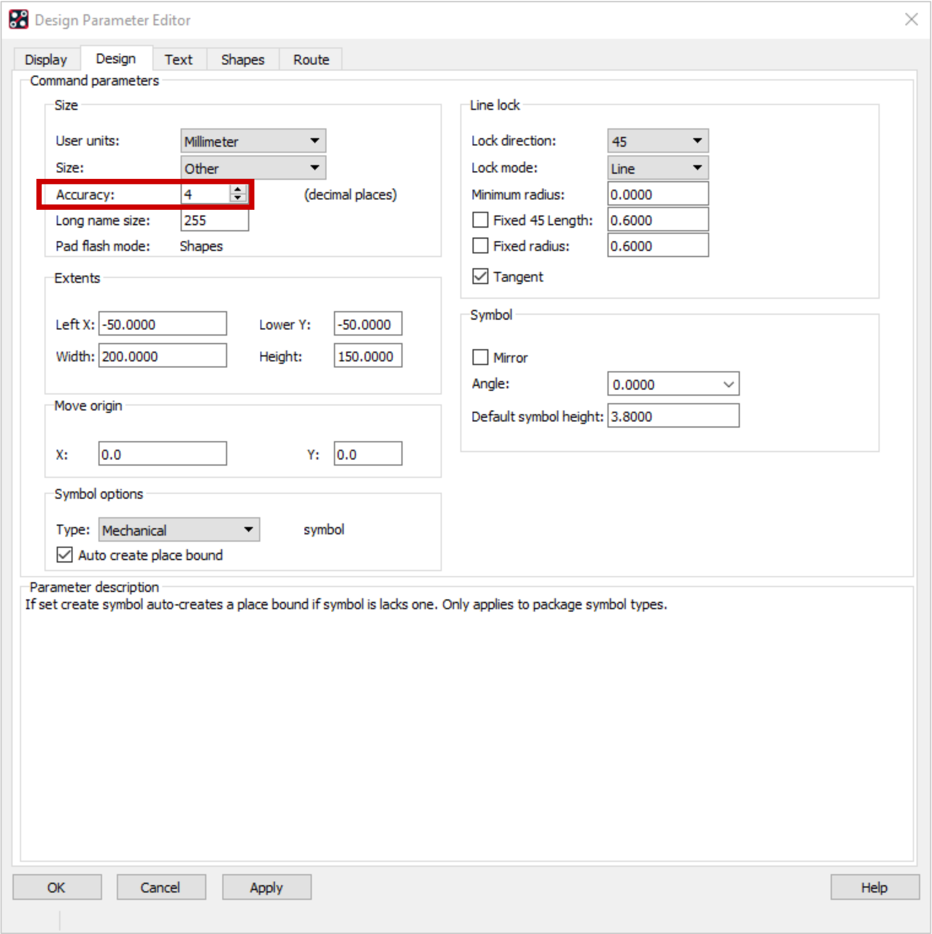
- **File > New > Mechanical Symbol**
- Enter **Outline** in Drawing Name field.
- **OK**
- Define units and dimensions of drawing page in dialog **Create a New Design**, which will appear automatically.
- **OK**





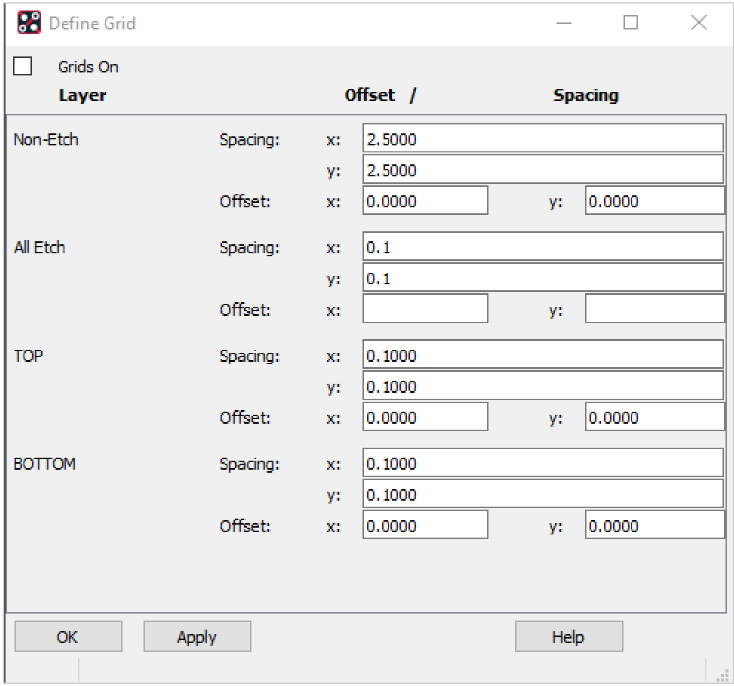
Lab: Board Symbol

- All other settings can be defined under **Setup > Design Parameters… > Design.**
- Precision of database should be high enough.



Lab: Board Symbol

- **Setup > Grids**
- In **Non Etch** section enter 2.5 for x and y.
- **OK** to close form.



Lab: Board Outline Generation

For our lab we assume that origin (0,0) of outline is the mounting hole on bottom left.

1. **Add > Line** from main menu.
2. Set active class and sub class to **Board Geometry / Outline.**
3. Please enter lines below in command line of editor and finish every line with **Enter**.

   x -30 10

   x -10 10

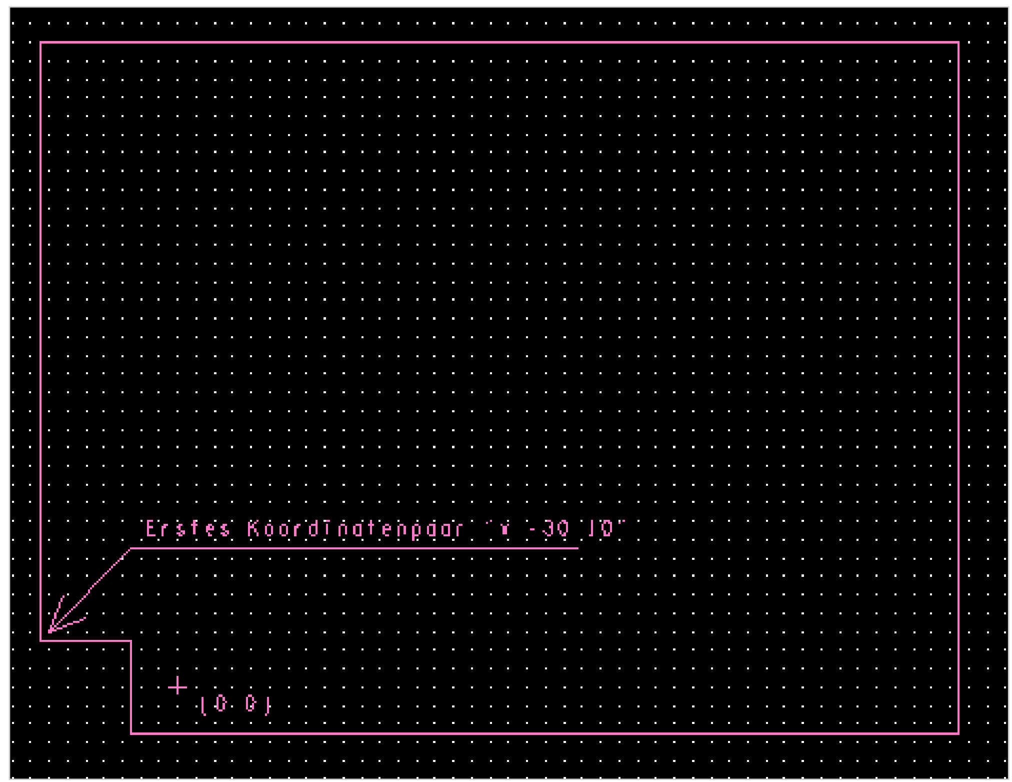
   x -10 -10

   ix 80

   iy 70

   ix -100

   iy -50
4. **RMB > Done.**
   Outline should look like illustrated in picture.



Tip

Absolut coordinates: x value value
Relative coordinates: ix or iy value

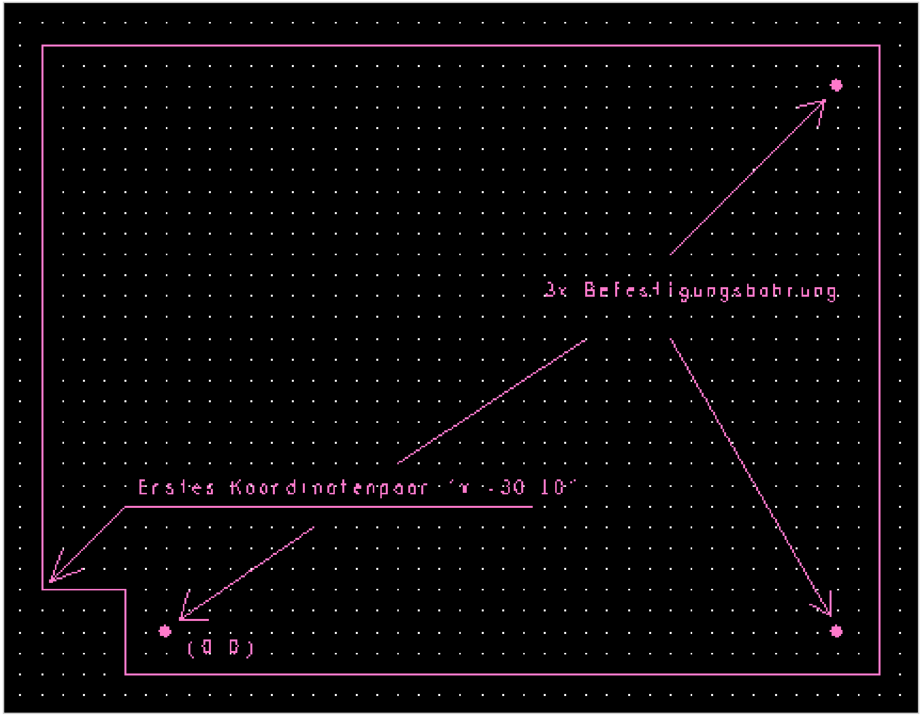
Lab: Mounting Holes

Second step is addition of mounting holes.

1. **Layout > Pin** from main menu or

   Click Browse button [...] in option window **Padstack** field
2. Select **Hole120**
   Editor shows the message using Hole120.pad,
   meaning it was possible to find pad in library.
3. Please enter data below one after another
   in command line and confirm each line
   with **Enter**.

   x 0 0
   x 60 0
   x 60 50
4. **RMB > Done.**

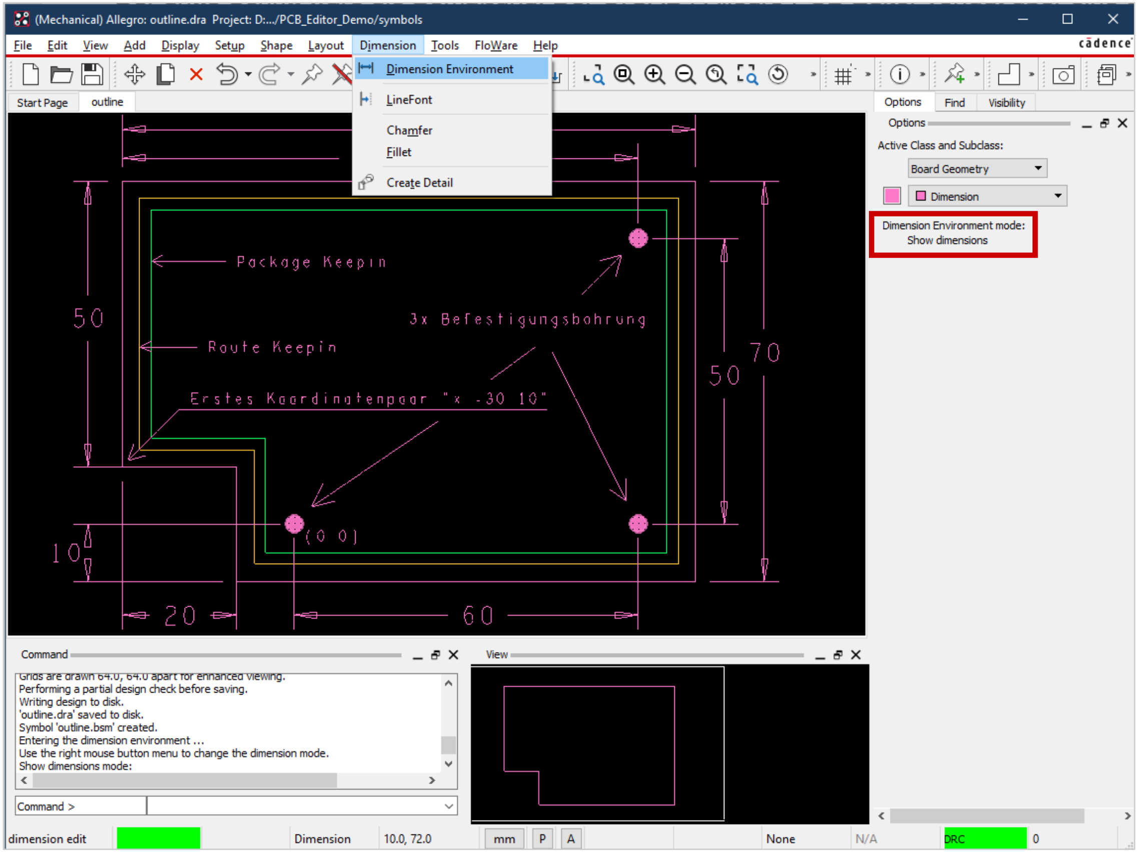


Lab: Dimensioning (I)

Third step is dimensioning. PCB Editor provides dynamic dimensioning. A change of contour triggers an automated update of dimensioning. Dimensioning can be entered in **PCB** or in **Symbol**. Usage is identical.

In PCB Editor you open:
Manufacturing > Dimension Environment

in Symbol Editor:
Dimension > Dimension Environment or via icon .

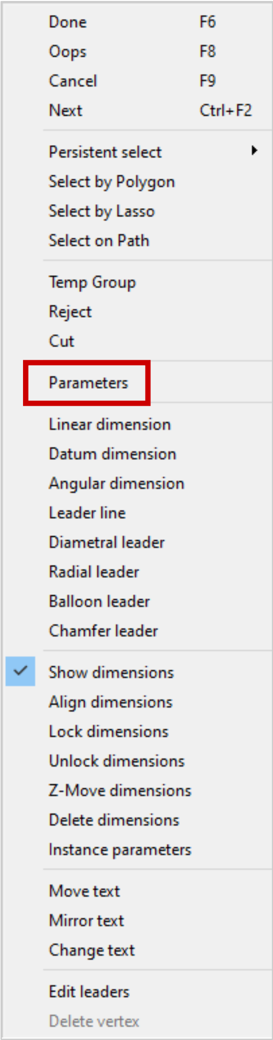


Lab: Dimensioning (II)

Necessary commands for dimensioning are accessible via a pop-up menu combined with **right mouse button (RMB)**.

Next, we will explain dimensioning with example of outline (mech. symbol).

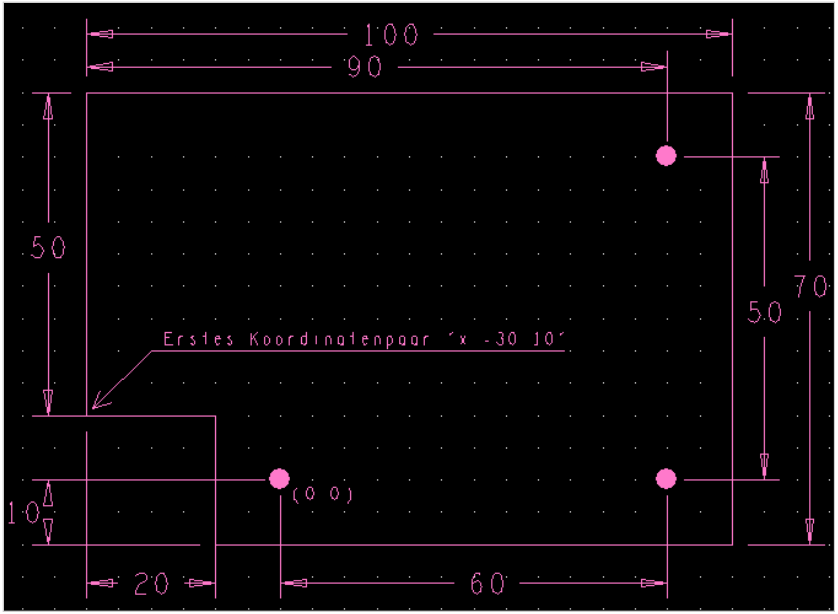
Dimensioning can be adjusted via parameters.

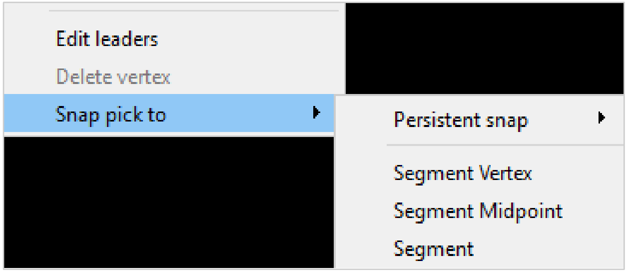


Lab: Dimensioning (III)

After parameter setting you can start with:

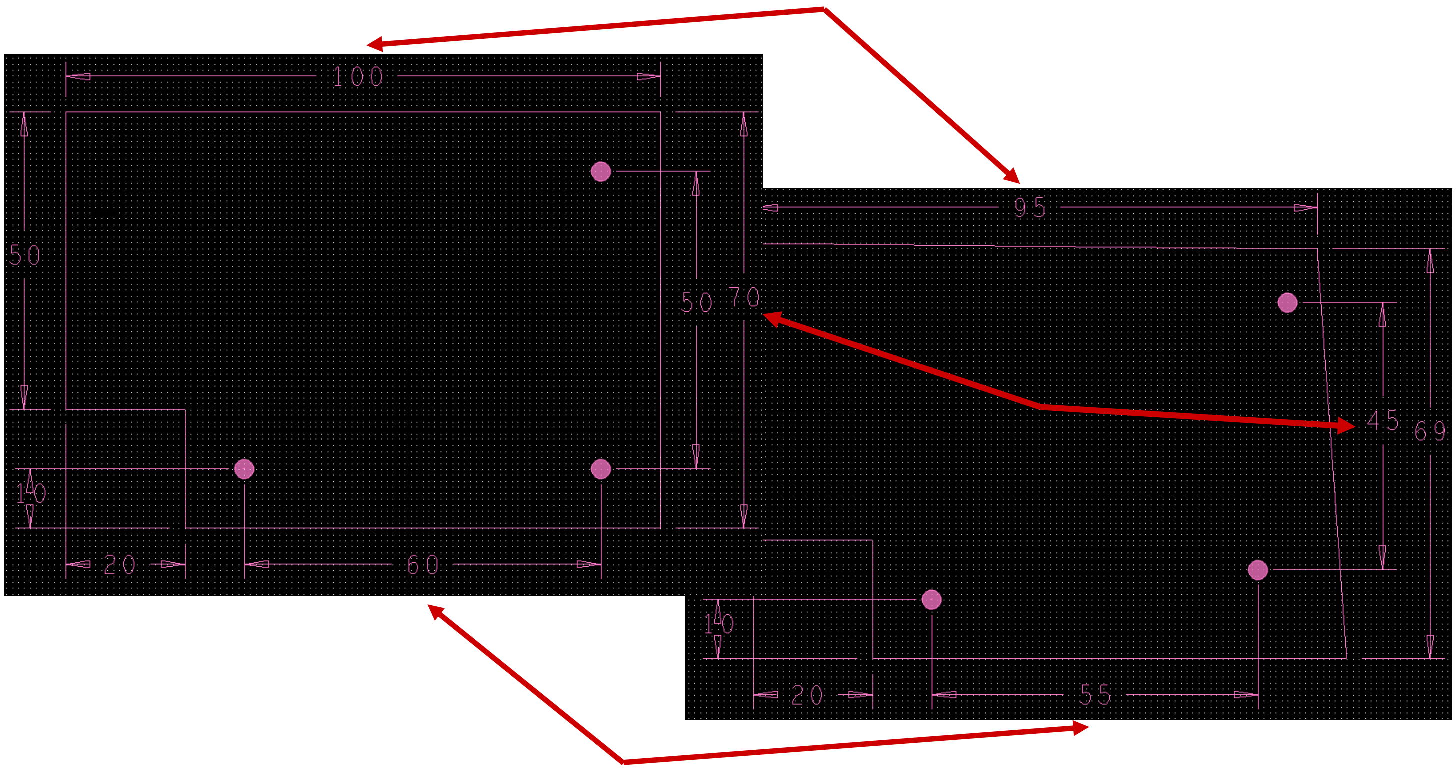
1. **RMB > Linear Dimension**
2. System is requesting to choose a location or an element for dimensioning.
3. Select top board edge (100) and place text.
4. Select right board edge (70) and place text.
5. Complete dimensioning like shown in the picture.
6. For a point to point dimensioning (drill hole bottom right -60) coordinates must be selected exactly (please note filter).
   You can use also snap function.
7. For non orthogonal dimensions an additional click is required to define direction of dimensioning (horizontal or vertical).
   Bottom left corner with dimensions 10 and 20.



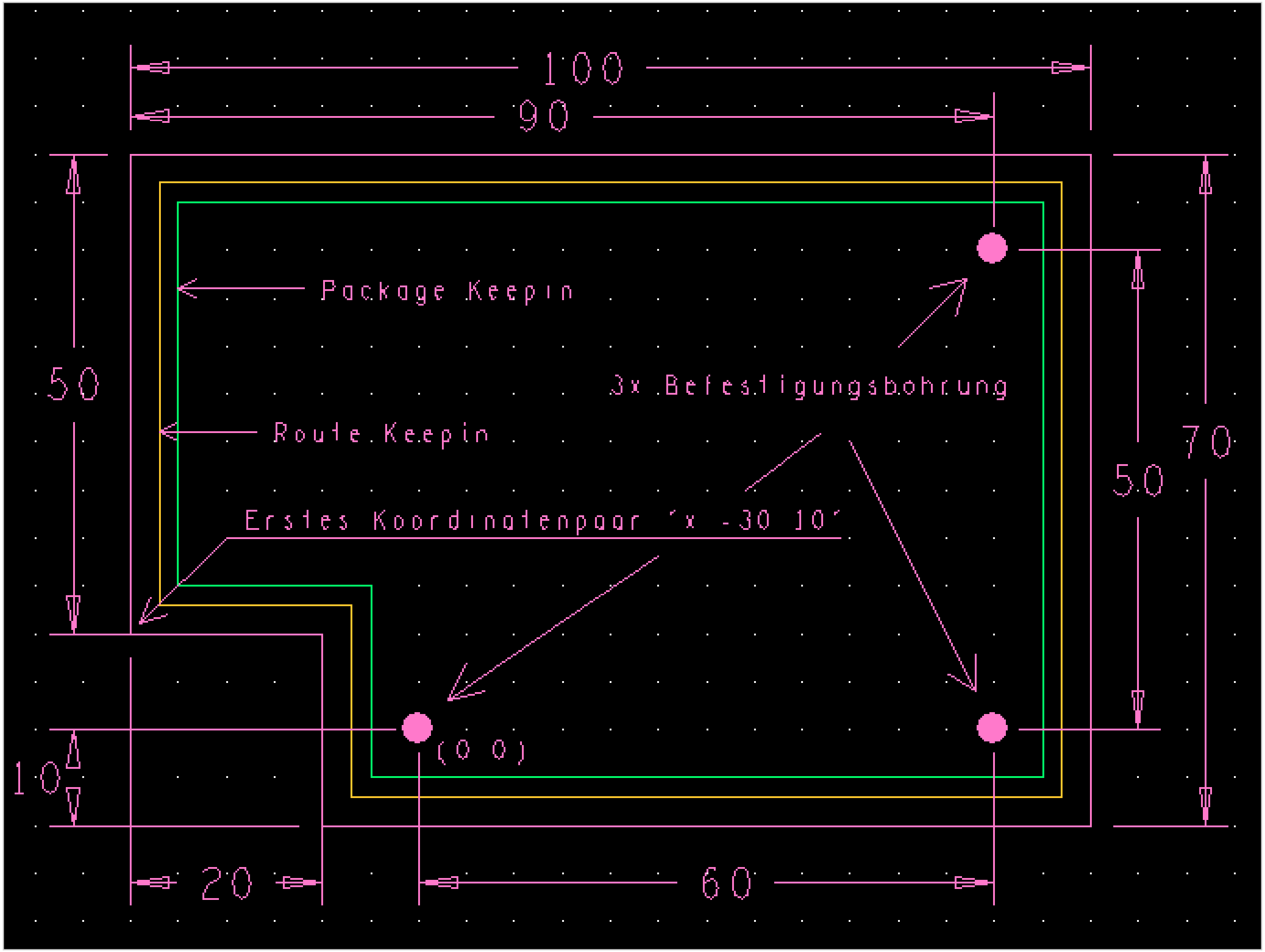


Lab: Dimensioning (IV)

8. To test dynamic behavior of dimensions please change position of bottom right mounting hole or top right corner of board outline.
9. Choose **Edit > Move** or icon ✥ and move mounting hole.
10. Choose **Edit > Vertex** and move top right corner.
11. In both cases dimensions are updated immediately.



Completed Board Symbol

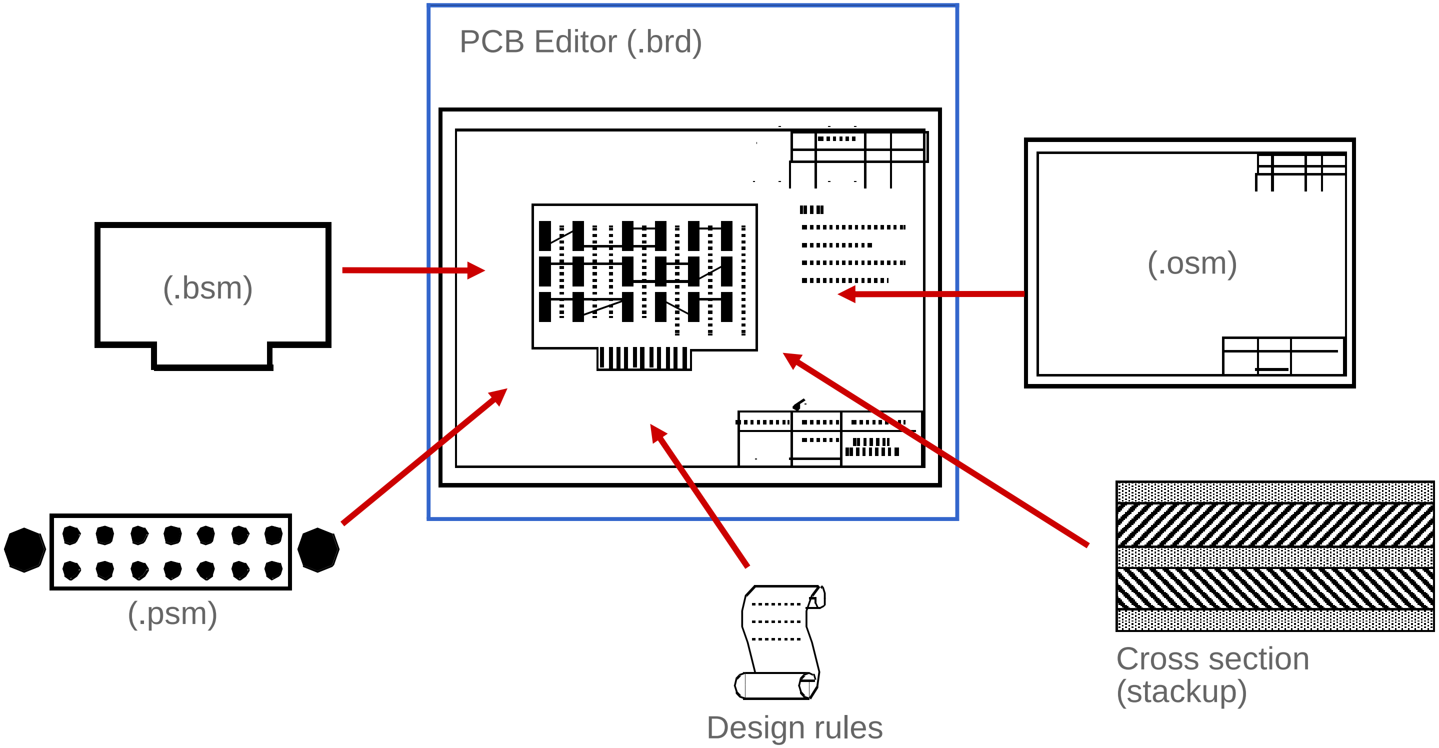


Master Board

Master board is a board template which contains next to board outline additional settings which are mandatory for PCB design. These are:

Layer stack, design rules, preplaced components, etc.

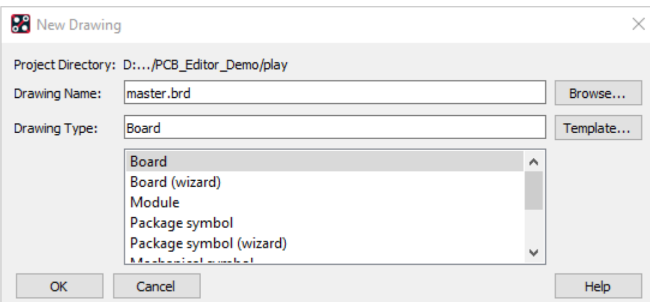
This method saves significant time and is improving quality because parameters are already good and will be reused.



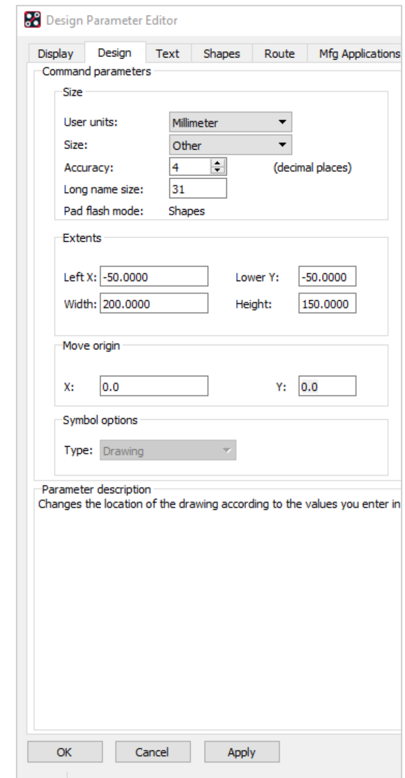
Lab: Master Board Setup

On next pages we will describe most important steps on how to create a master board. These steps apply to any other board too.

1. **File > New** from main menu.
2. Enter **master** in Drawing Name field.
3. Select Drawing Type **Board.**
4. **OK.**



5. **Setup > Design Parameters… > Design**
6. Please change values like in drawing parameter box on right side.
7. **OK.**



Tip

Ensure that Drawing Extents are large enough to cover additional elements like drawing frames. Accuracy should be chosen high to ensure that shapes and fine traces have right resolution.

Lab: Master Board Layer Stack

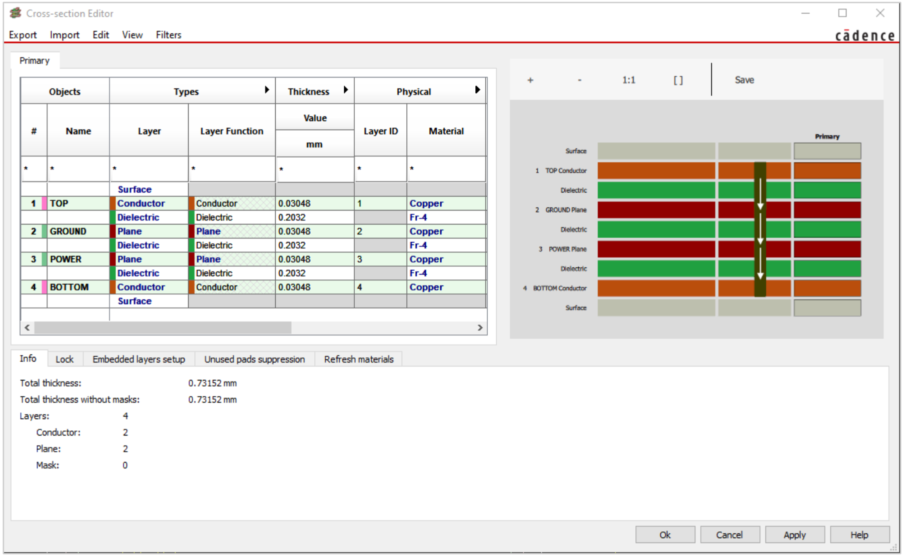
Settings for layer stack can be reached via **Setup > Subclasses > Etch** or 

This menu allows to add or remove electrical layers (routing layers).
RMB > **Add Layer / Remove Layer**.

Subclass Name of TOP and BOTTOM is default and cannot be changed. All additional layers can have any individual (unique) name. Chosen name appears also in Option Visibility window.

With **Type** it is possible to chose between Conductor (routing layer), Dielectric and Plane (power layer).

Negative Artwork dictates type of output. Normal signal layers (conductor) are usually positive. Plane layers can be defined positive and negative.



Tip

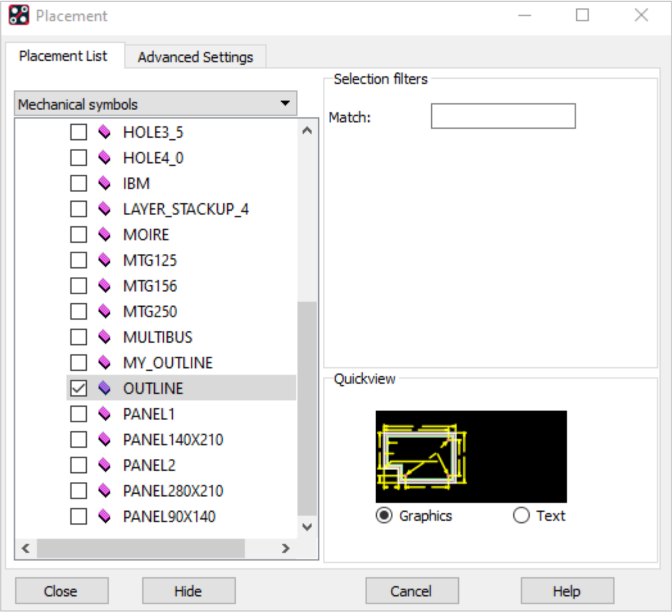
It is only possible to delete additional layers if these layers no longer contain any data.

Lab: Master Board, Board Symbol

Like mentioned before, it makes sense to add a predefined board symbol (mechanical symbol) into a Master board for repeating designs. This makes design process effective and reliable.

With next steps, we load board symbol into open master board **master.brd**.

1. **Place > Manually** from main menu, a placement box (right side) appears.
2. Select under **Advanced Setting** tab both options, **Database** and **Library.**
3. Expand mechanical symbols in placement list and select **outline** (self defined symbol).
4. Enter in command line **x 0 0** and **Enter**.
5. **RMB > Done.** Mechanical symbol is placed.
6. **File > Save As.** A **master.brd** file will be saved.
7. Do not close PCB Editor!



Tip

At this point you can also preplace components (package symbols), format symbols and define other pre settings for grid, colors and design rules.
Design rules are dealt with from page 74 onwards.

Final Statements

As mentioned in the beginning, this tutorial is a quick start guide and not a training and should not replace one.

Quick start should give you an overview of functionality in OrCAD flow and should enable you to make first independent steps. Main reason is to empower you to judge, if OrCAD flow meets your needs.

Independent of this quick start we recommend a training to enable you to use full performance and highest efficiency for your daily tasks.

Trainings are scheduled and delivered by Cadence and FlowCAD on a regular base. For details please visit websites from Cadence and FlowCAD:

www.cadence.com

www.FlowCAD.de/Training

Appendix

System Requirements (Full Version 17.4)

Operating System	Windows 10 (64-bit) Professional, Windows 2012 Server (All Service Packs); Windows Server 2012 R2; Windows 2016 Server
Hardware	Intel® Core™ i7 4.30 GHz or AMD Ryzen™ 7 4.30 GHz with at least 4 cores 16 GB RAM 50 GB free disk space (SSD drive is recommended) 1920 x 1200 display resolution with true color (at least 32 bit color) A dedicated graphics card supporting OpenGL, minimum 2 GB (with additional support for DX11 for 3D Canvas) Dual monitors (for physical design) Broadband Internet connection for some services

Properties of Full Version

General

- Grid imperial or metric
- Easy to create or edit libraries
- Forward-backward annotation
- Cross-probing between design entry (Capture) and layout (PCB Editor)

Schematic editor Capture

- Maximum workspace up to 11.430 x 11.430 mm
- Multiple Designs as part of one project
- Hierarchical structures with automated synchronization
- Automated Reference designator assignment
- Electrical Design Rules Check (configurable)
- Configurable automated drawing border and title block
- Export of different netlist formats
- TCL interface

Usage Concept

PCB Editor allows command entry in five different ways.

1. **Command line**
2. **Pull-down menu and context-sensitive Pop-up menus (Pre-Selection Mode)**
3. **Icon**
4. **Short key**
5. **Strokes**

All commands are accessible via command line.

Pull-down menus contain almost (few exceptions) all commands who are available via command line.

Icons provide most important and used commands.

Short keys and strokes are user configurable. Cadence default settings can be modified or extended. Icons can be bundled in new groups.

In Pre-Selection Mode we use context sensitive menus. Meaning, dependent on selected object system provides only functions related to this object.

Note: RMB = right mouse button

Contact us / Kontakt zu FlowCAD

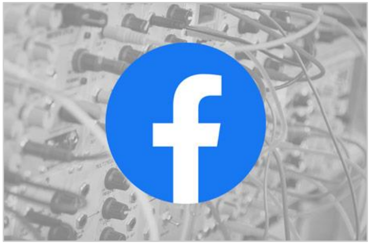
Please do not hesitate to contact us.
Für weitere Fragen und Informationen stehen wir gerne zur Verfügung.

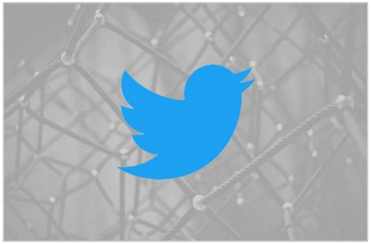
FlowCAD Deutschland
Mozartstr. 2
85622 Feldkirchen bei München
T +49 89 45637-770
info@FlowCAD.de

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

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!

FlowCAD