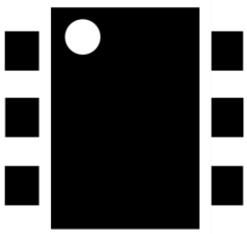


# Printed Circuit Board Workbench for FreeCAD
# PCB-FreeCAD

marmni (marmni@onet.eu)

https://github.com/marmni/FreeCAD-PCB

https://sourceforge.net/projects/eaglepcb2freecad/

https://www.freecadweb.org/


https://www.sqlalchemy.org/


https://www.python.org/


https://pypi.org/project/PySide/

# Spis treści

# GENERAL INFORMATIONS

# LICENCE

# INTRODUCTION

Printed Circuit Board Workbench for FreeCAD. Workbench allows you to:

1. Importing boards created in various dedicated PCB softwares. Layers/colors are supported. Supported softwares:
   - Eagle (*.brd),
   - FreePCB (*.fpc),
   - gEDA (*.pcb),
   - KiCad (*.kicad_pcb),
   - IDF v2/v3.

2. Creating and exporting boards to various formats. Supported formats:
   - Eagle (*.brd),
   - FreePCB (*.fpc),
   - gEDA (*.pcb),
   - KiCad (*.kicad_pcb),
   - IDF v2/v3.

Workbench supports 3D models saved in one of the following formats: STP/IGS

## Requirements

FreeCAD-PCB require FreeCAD in version 0.18 (or newer) and Python **2.7** (or newer). Module was tested on Windows and GNU/Linux.

## Supported softwares

| Soft name | PCB | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|
| | | Holes/Vias | Parts | Border | Measures | Soldermask | Keepout layers | Paths | Pads | Soldermask ARC | PCB round corners | Annotations |
| Eagle | brd | | | | | | | | | | | |
| gEDA | pcb | | | | | | | | | | | |
| FreePCB | fpc | | | | | | | | | | | |
| KiCad | kicad_pcb | | | | | | | | | | | |
| FidoCadJ | fcd | | | | | | | | | | | |
| Razen | rzp | | | | | | | | | | | |
| IDF v2 | idf | | | | | | | | | | | |
| IDF v3 | idf | | | | | | | | | | | |
| IDF v4 | | | | | | | | | | | | |
| HyperLynx | HYP | | | | | | | | | | | |

Yes

No

Never

In progress

Future

# INSTALLATION

There are two method to install workbench: manually and by FreeCAD-addons manager. Second solution is recommended.

## Manual installation

Unpack downloaded zip file from github/sourceforge and copy extracted folder to:

- **GNU/Linux**

  On GNU/Linux distributions better do not keep PCB workbench folder under standard FreeCAD installation path (often under /sys path). This is connected with root permissions. Better idea is to keep it under /home directory.

  Go to your user directory '/home/**userName**' and display all hidden folders. Search for folder ".FreeCAD". Under this directory you should find subfolder 'Mod'

  ---

  */home/**userName**/.FreeCAD/Mod*

  ---

  Replace **username** with our user name

  Next change read/write permissions to 777 (also for subfolers).

  ---

  *chmod 777 -R PCB*

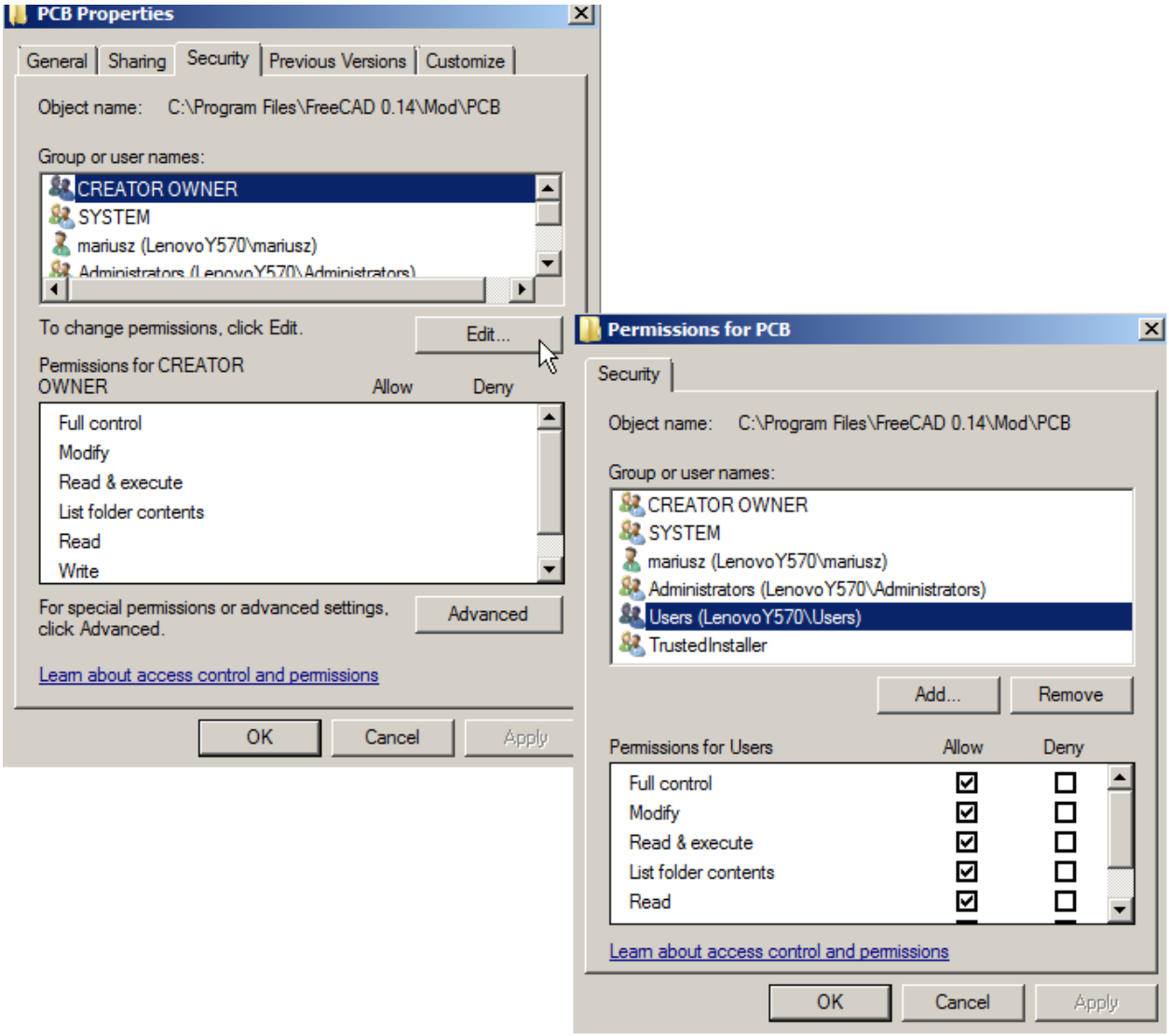
  ---

- **Windows**

*C:\Users**userName**\AppData\Roaming\FreeCAD\Mod\PCB*

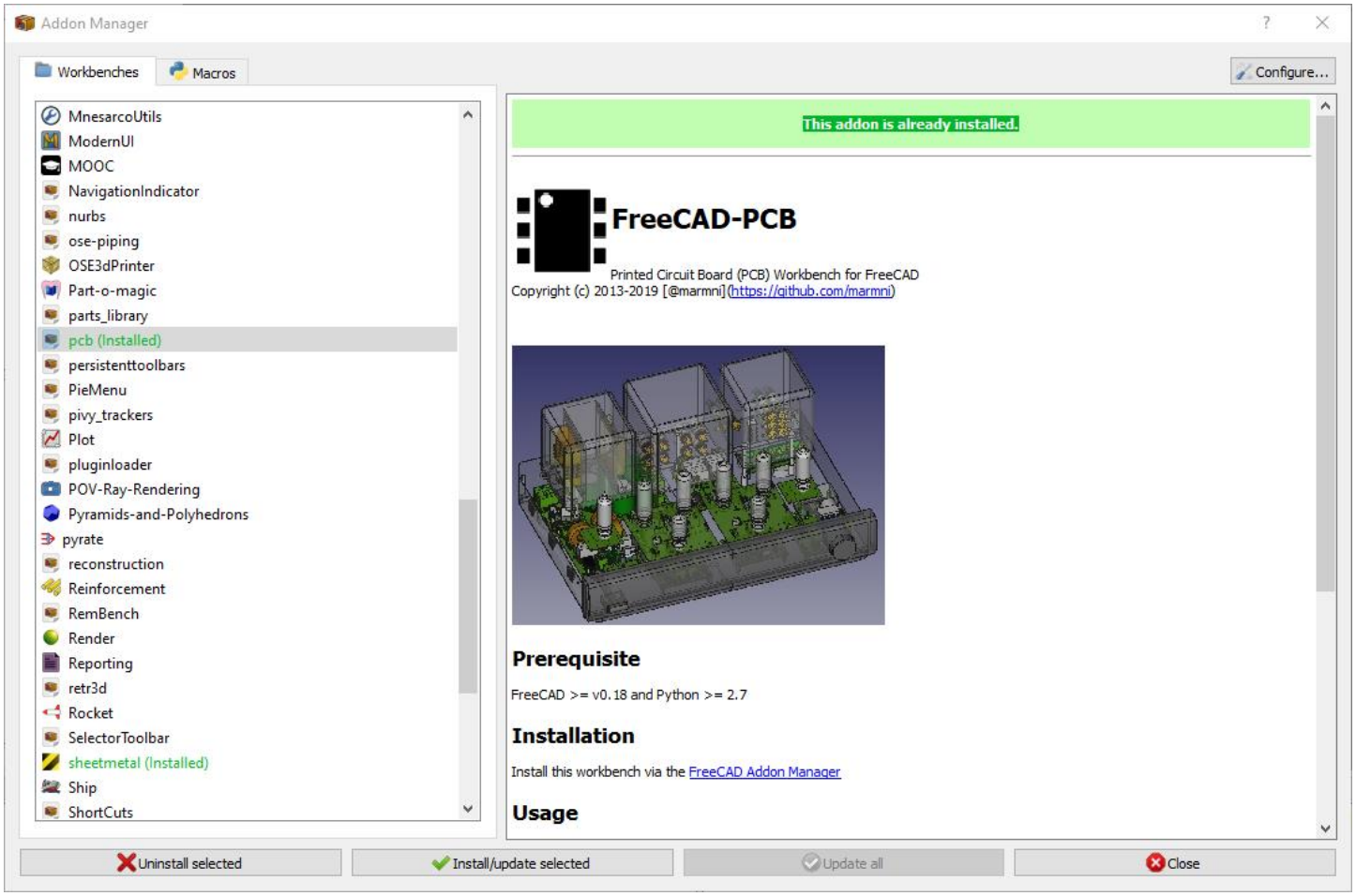
Replace **username** with our user name.

It is recommended also to change read/write permission for all users. Click right button on PCB folder and select Properties → Security → Edit → Users and mark all checkboxes under 'Allow' option.



## Addon manager

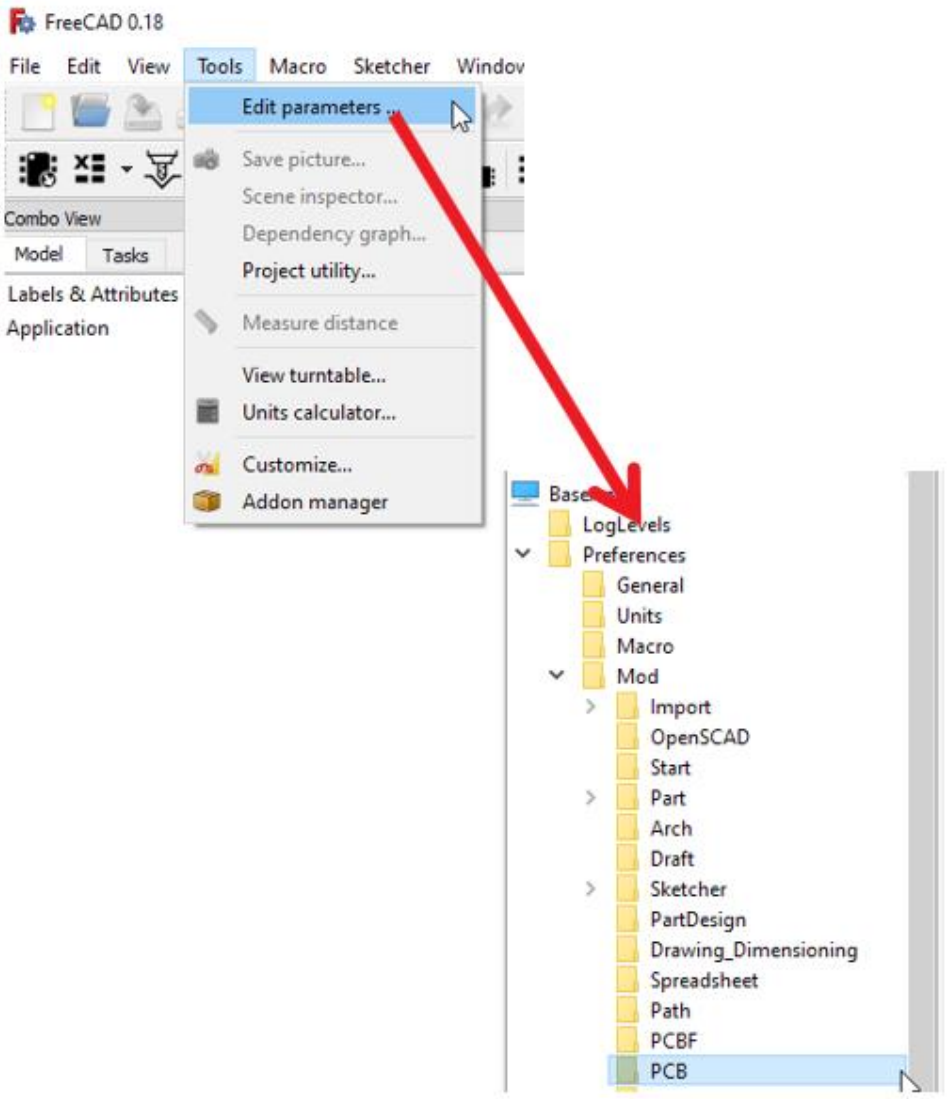
FreeCAD-addons is a standard part of FreeCAD which allows you to automatically install new workbenches/macros for FreeCAD. You will find it in the main menu Tools -> Addon manager




It is recommended to use the add-ons manager instead of manual installation. The manager also allows you to update installed modules.

# CONFIGURATION

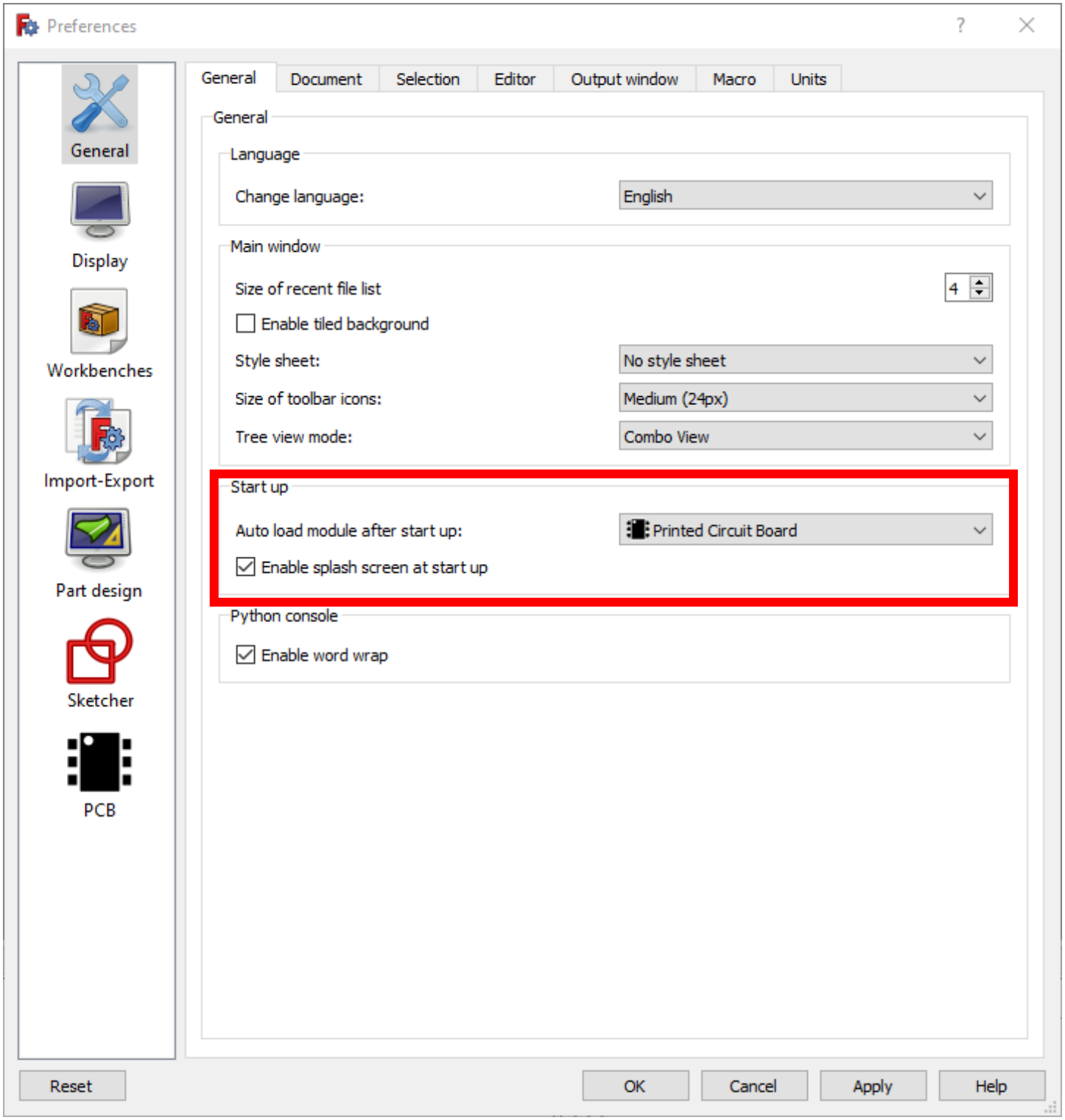
Most of variables/parameters are stored in FreeCAD configure files. You can find a parameter editor in the main menu.




**Do not change anything in file 'PCBconf.py'!**

## Setting PCB module as main workbench

There is a possibility to set PCB module as main workbench. To do this choose General tab (Edit -> Preferences). Under this tab you should find 'Start up' section, where you can set which workbench should be loaded after FreeCAD start.



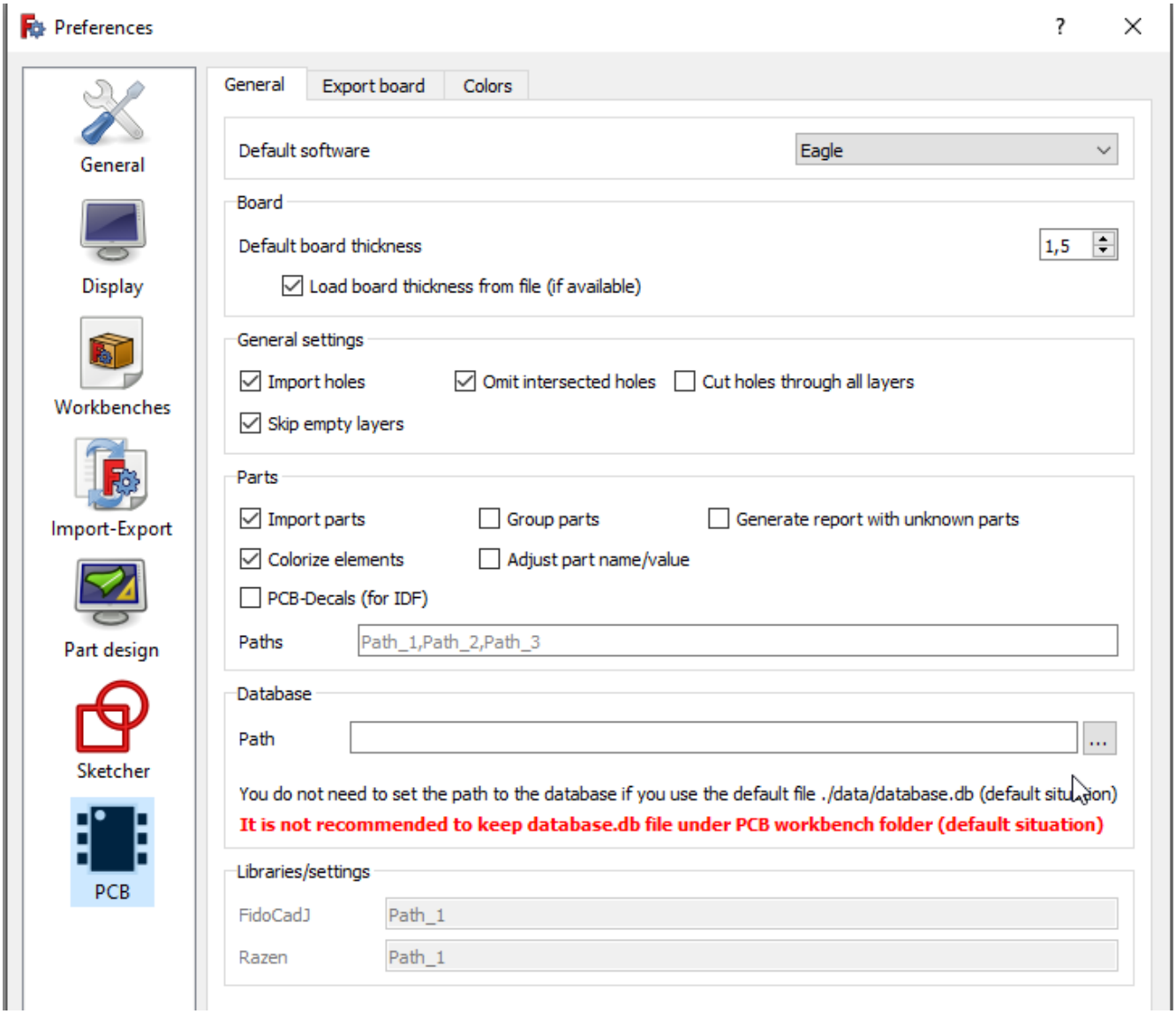
# CUSTOMIZING WORKBENCH

Workbench is fully configurable - you can set various variables which are directly connected with importing/exporting PCB boards. All setting are automatically stored in FreeCAD so you need to set them only once (of course you can change them also whenever you want).

In main menu choose Edit → Preferences - > PCB.

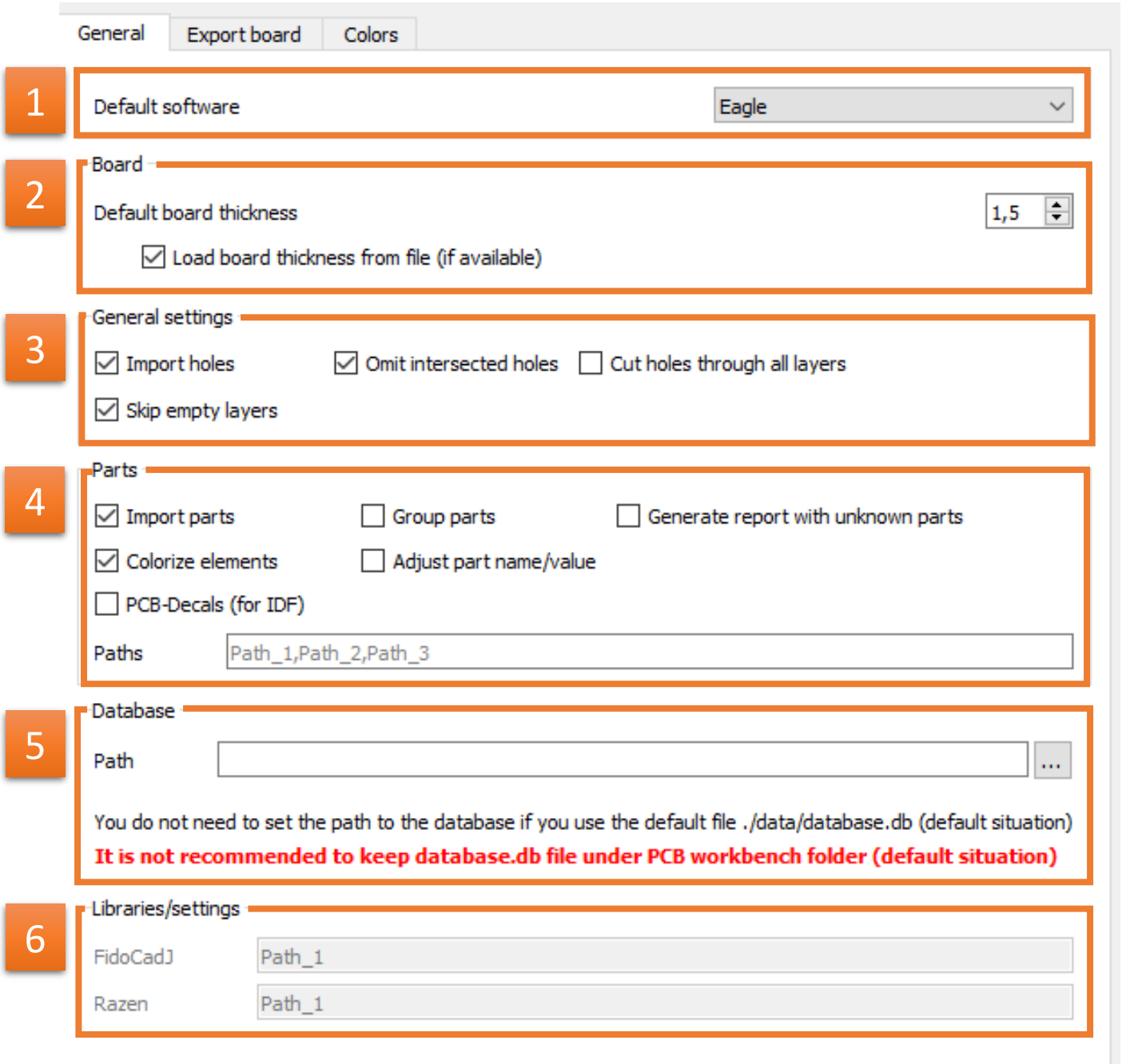
In the PCB section you can find all the configuration settings that are included in three tabs:

- General
- Export board
- Colors



## General

In this tab all settings are arranged under six sections.



1 Set default software which you are using.

2 Set default boart thickness (parameter can be changed even after importing the board)

3 **Import holes** – import holes from file (if checked)

**Omit intersected holes** - to avoid problems during loading PCB board this option should be always checked

**Cut holes through all layers** – it is possible to show/hide holes in layers (for example paths/pads, parameter can be changed even after importing the board)

**Skip empty layers** – this option decrease time necessary to generate 3D representation of the PCB board – empty layers will not be generated

4 **Import parts** – import or skip 3D models of the parts

**Group parts** – group imported 3D models in categories

**Generate report with unknown parts** – if 3D representation for imported package will not be recognize you can generate report (txt file) which will contain missing 3D models

**Colorize elements** – import colorized 3D models (if selected) or gray scale models (if option is not selected)

**Adjust part name/value** -

**PCB-Decals** – check this option if you will import IDF files

**Paths** – add here a path under which you are storing 3D models. Do not change anything (leave empty) if you are using standard localization (PCB/parts) . Separate different paths by comma

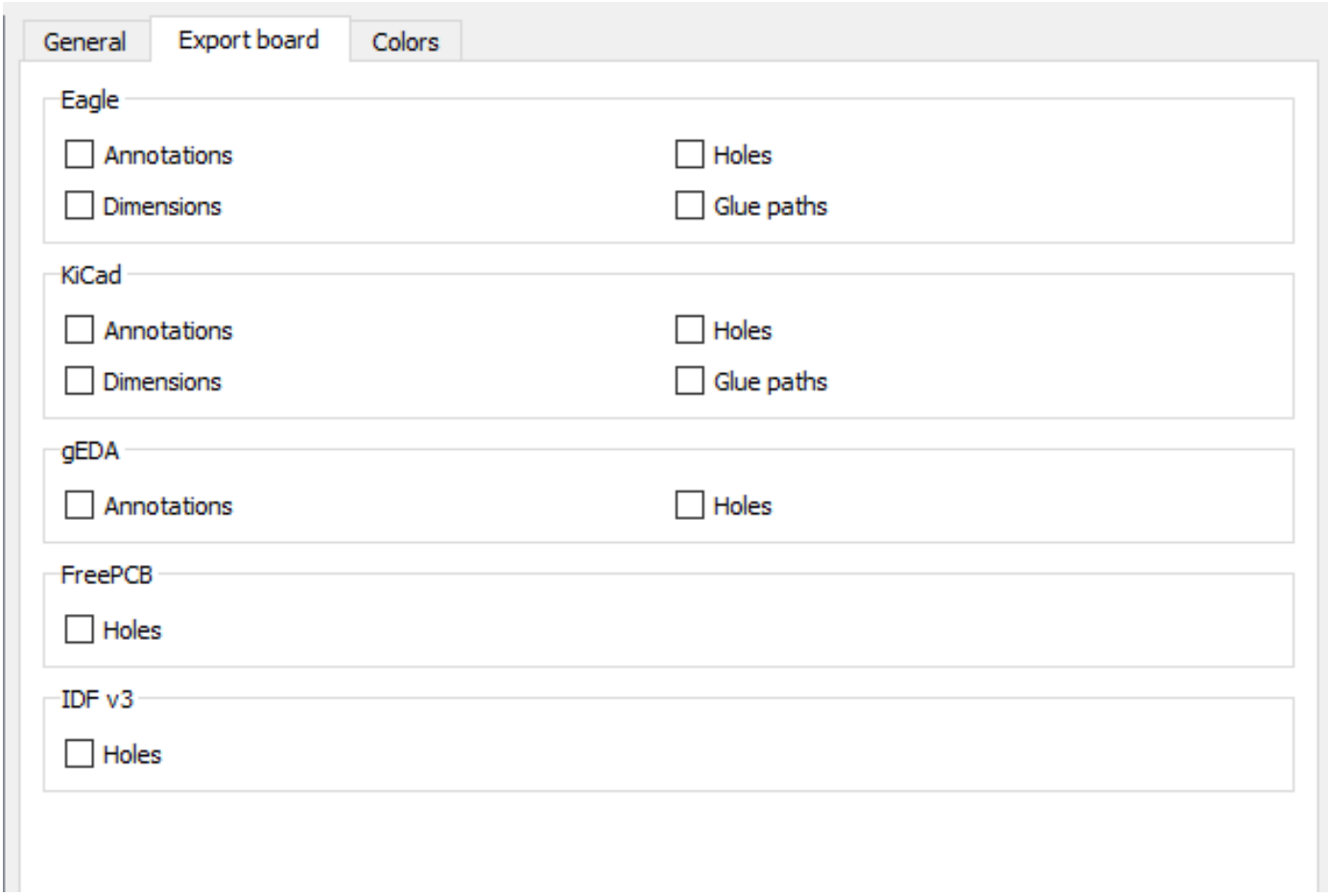
5 **Path** – path to database.db file (only one). Do not change anything (leave empty) if you are using standard localization (PCB/data/atabase.db)

**It is not recommended to keep database.db file under PCB workbench folder (default situation)**

6 Libraries – for future

## Export board

In this section you can set the default layers that will be included when exporting the board to one of the supported formats.



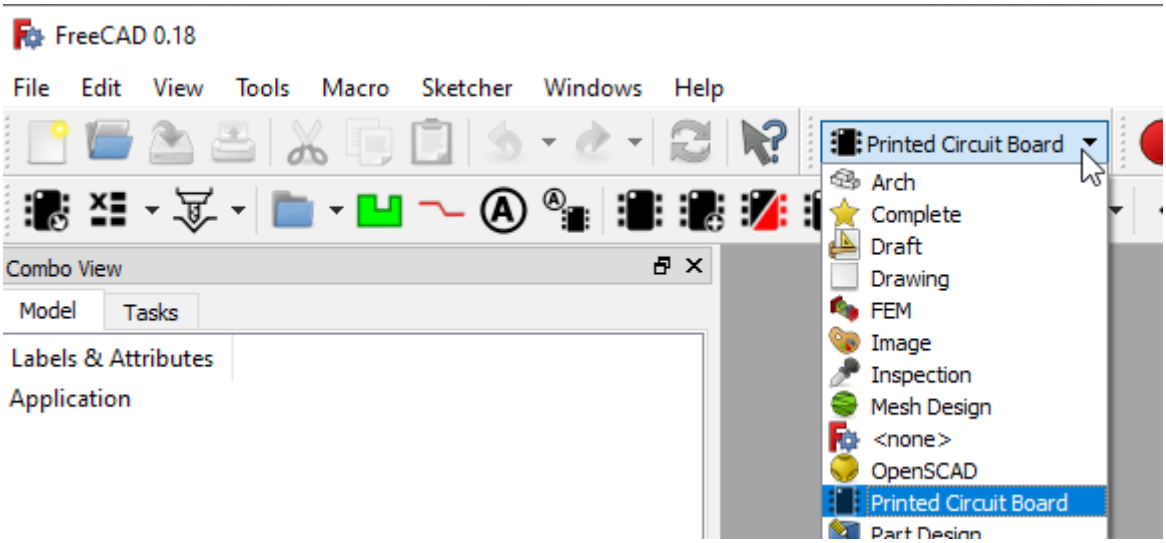
## Colors

Default colors for imported layer.

# ACCESSING THE WORKBENCH

There are two methods to access to the PCB workbench:

- In one of the available toolbars locate drop down list and choose ’Printed Circuit Board’.




- From top menu bar choose View → Workbench → Printed Circuit Board.

# MENU BAR

There are no menu bars dedicated for PCB workbench.

# TOOLBARS

Two special toolbars are available:

- PCB View.
- PCB Settings.

This section describes the various icons available in mentioned toolbars.

## PCB Settings toolbar

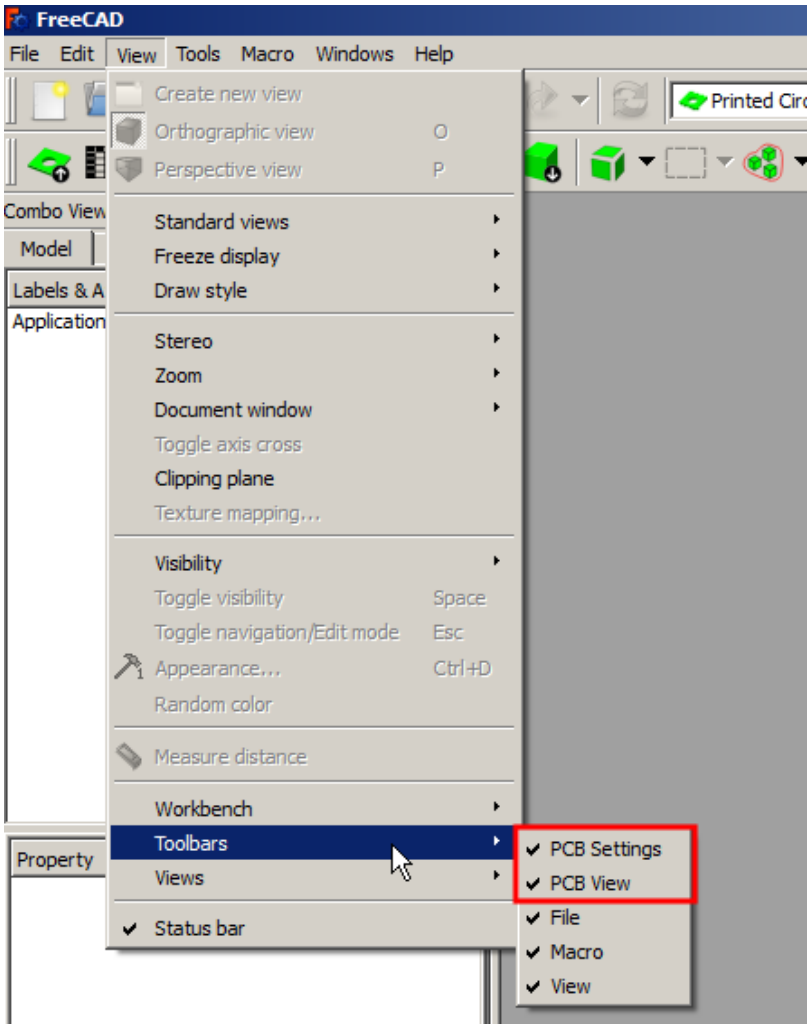
| | Option | Description |
|---|---|---|
| | Export PCB | Check Export board section |
| | Export BOM | Check Export Bill Of Materials (BOM) section |
| | Centroid | Check centroid section |
| | Export hole locations | Check Export hole locations section |
| | Export hole locations report | Check Export hole locations report section |
| | Create drilling map | Check Create drilling map section |
| | Create PCB | Check Create PCB section |
| | Create glue path | Check Create glue path section |
| | Add annotation | Check Add annotation section |
| | Store Name/Value as parm | |
| | Assign models | Check Assign models section |
| | Add model | Check Add model section |
| | Update models | Check Update models section |
| | Download models | Check Download models section |
| | Generate models | Check generate models section |

| | Option | Description |
|---|---|---|
| | Explode | Check Explode section |
| | Create constraint area | Check Create constraint area section |
| | Bounding box | Check Bounding box section |
| | Section cuts | Check Section cuts section |

## PCB View toolbar

| | Option | Description |
|---|---|---|
| | Change display mode to Shaded | Check Display modes section |
| | Change display mode to Flat Lines | Check Display modes section |
| | Change display mode to Wireframe | Check Display modes section |
| | Change display mode to Internal View | Check Display modes section |
| | Layers settings | Check Layers section |
| | Cut holes through all layers ON/OFF | Check Cutting holes through all layers section |
| | Cut to board outline | Check Cut to board outline section |
| | Show signals | Check Signals marking section |
| | Group/Ungroup models in 'Parts' folder | Check Grouping parts section |
| | 3D rendering: export to Kerkythea | Check Kerkythea section |
| | 3D rendering: export to POV-RAY | Check POV-RAY section |
| | Instructions | |

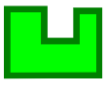
## Displaying toolbars



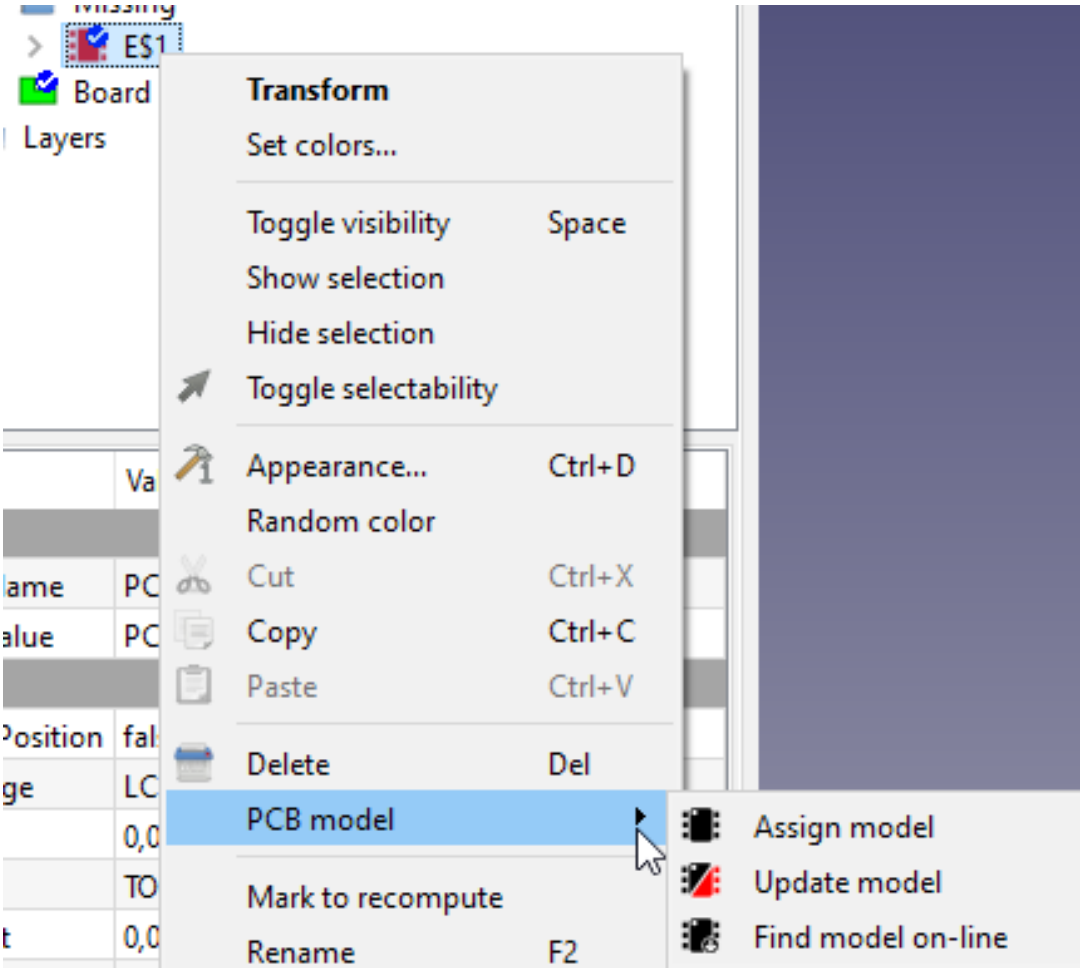

When mentioned toolbars are not displaying automatically after choosing PCB workbench in the main window, you need to do it manually. From top menu bar choose View → Toolbars and mark toolbars from Printed Circuit Board workbench.

# SPECIFICATION TREE

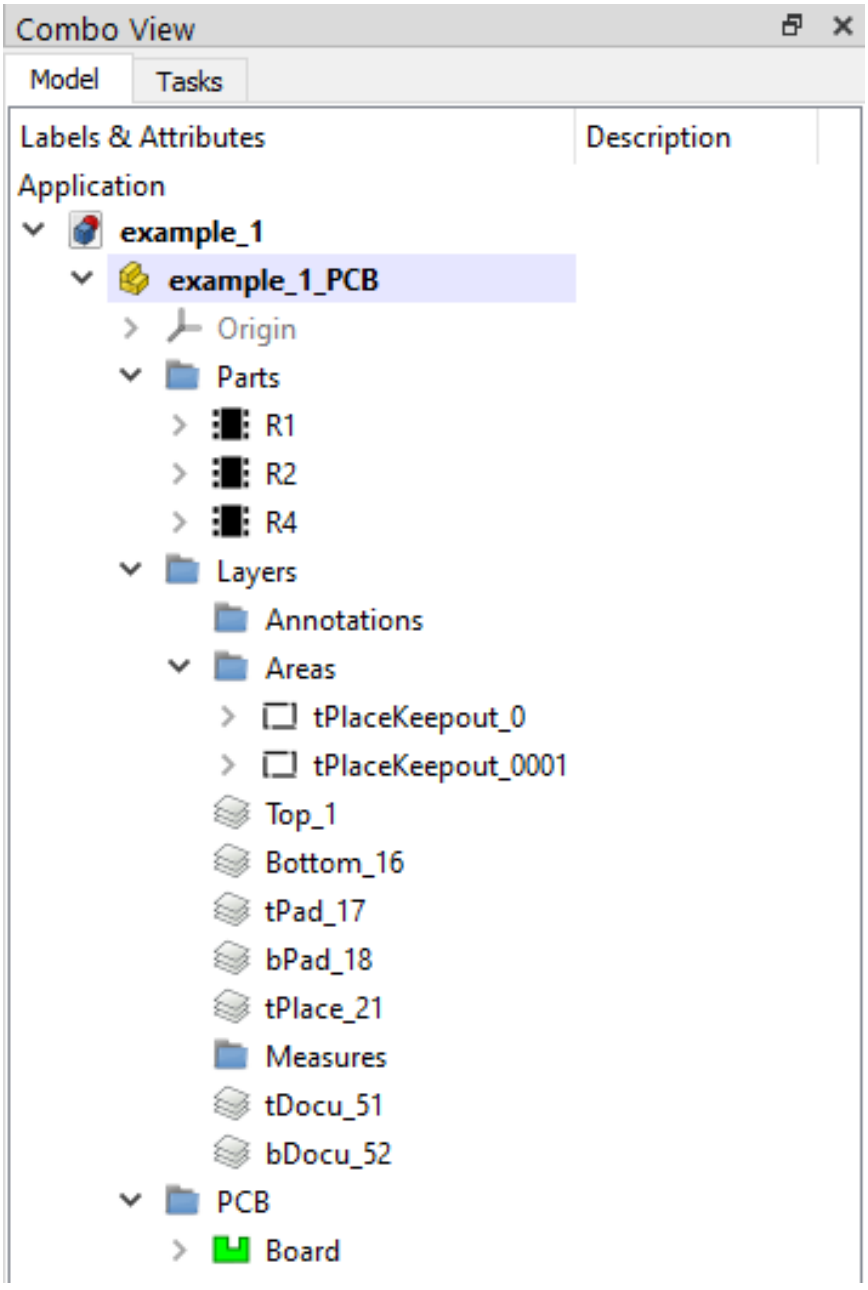
There are several types of objects directly related to the PCB workbench. You can see them in the "Combo View" and recognize them by the corresponding icons. For some of them specific context menu is available.

| | Object | Context menu |
|---|---|---|
|  | Explode objects | **Edit**. For more details check Explode section |
|  | 3D representation found in database | **Update model**. For more details check Updating models section<br><br>**Placement model.** Change/update offset coordinates (in databease) |
|  | The 3D representation was not found in the database | **Assign model**. For more details check Assign models section<br><br>**Update model.** For more details check Updating models section<br><br>**Find model on-line**. For more details check 3D models section |
|  | Board | ---------- |
|  | Constraint area | ---------- |
| | | |

| | Object | Context menu |
|---|---|---|
|  | Layer | ---------- |
|  | Annotation/Object Name/Object Value | ---------- |
|  | Glue path | ---------- |

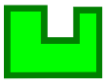


**Context menu for specific objects is only available in the PCB environment.**

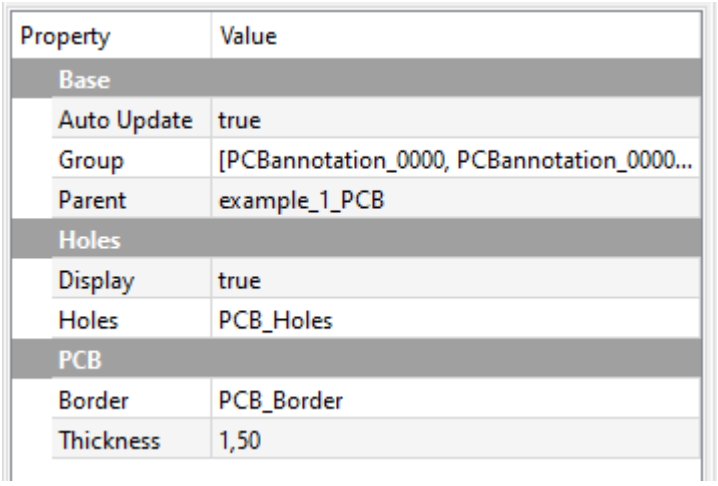


# OBJECTS PROPERTIES

Each object created in PCB workbench has unique parameters that can be set in the Property View (View or Data tab).



## Board



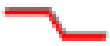

**Group:** all objects which are directly connected with board

**Display Holes**: show/blank holes

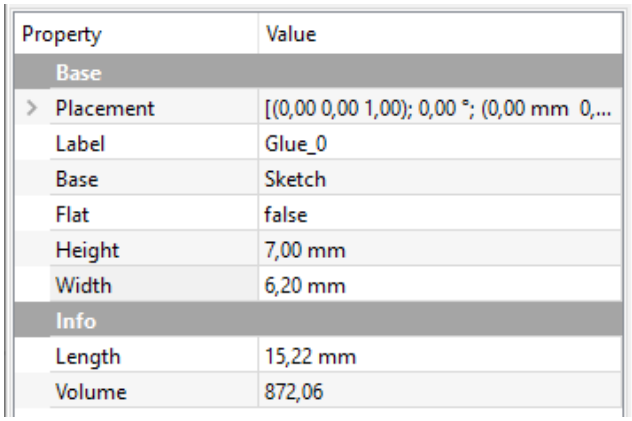
**Holes**: reference to sketch that containing holes

**Border**: reference to sketch that containing board outline

**Thickness**: board thickness



## Glue path




**Base**: reference to sketch that containing glue path shape

**Flat**: if this parameter is set to True, object will ignore Width/Height parameters

**Height**: glue path height

**Width**: glue path width

**Length**: glue seam lenght (for specific W/H)

**Volume**: for specific W/H

# Part model founded in database

**Part Name**: reference to part name object

**Part Value**: reference to part value object

**Keep Position**: part will ignore changes in correction values if this value will be set to True

**Package**: 3D model name, parameter disabled for editing

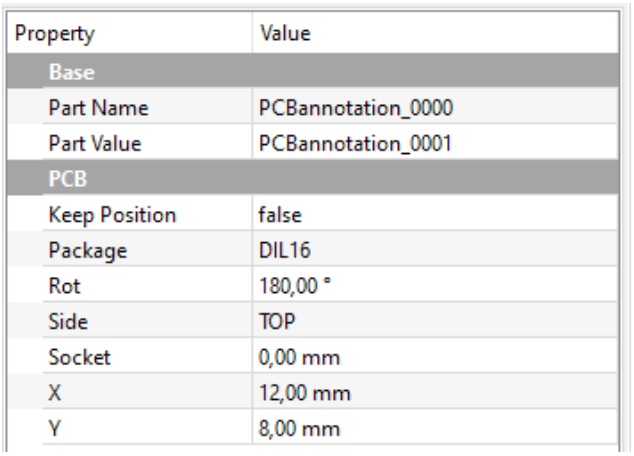
**Rot**: rotation value around Z axis

**Side**: part position on board (top/bottom side)

**Socket:** socket height / model position in Z direction

**X**: model position in X direction

**Y**: model position in Y direction

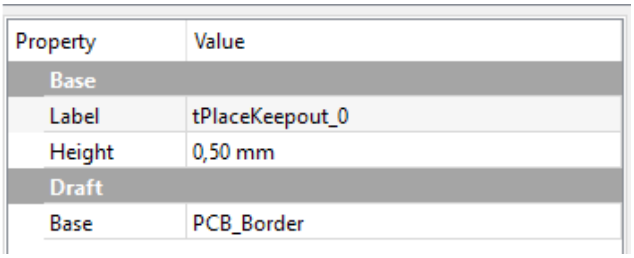


| Property | Value |
|---|---|
| **Base** | |
| Part Name | PCBannotation_0000 |
| Part Value | PCBannotation_0001 |
| **PCB** | |
| Keep Position | false |
| Package | DIL16 |
| Rot | 180,00 ° |
| Side | TOP |
| Socket | 0,00 mm |
| X | 12,00 mm |
| Y | 8,00 mm |

# Constraint area

**Height**: area height, parameter available only for some constraints areas type (on TOP/BOTTOM side)

**Base**: reference to sketch that containing area outline



| Property | Value |
|---|---|
| **Base** | |
| Label | tPlaceKeepout_0 |
| Height | 0,50 mm |
| **Draft** | |
| Base | PCB_Border |

## Part model not founded in database

**Part Name**: reference to part name object

**Part Value**: reference to part value object

**Keep Position**: part will ignore changes in correction values if this value will be set to True

**Package**: 3D model name, parameter disabled for editing

**Rot**: rotation value around Z axis

**Side**: part position on board (top/bottom side)

**Socket:** socket height / model position in Z direction

**X**: model position in X direction

**Y**: model position in Y direction

| Property | Value |
|---|---|
| **Base** | |
| Part Name | PCBannotation_0010 |
| Part Value | PCBannotation_0011 |
| **PCB** | |
| Keep Position | false |
| Package | LCC20 |
| Rot | 0,00 ° |
| Side | TOP |
| Socket | 0,00 mm |
| X | 57,00 mm |
| Y | 9,00 mm |

## Explode objects

**Active**: turn of/off explode effect

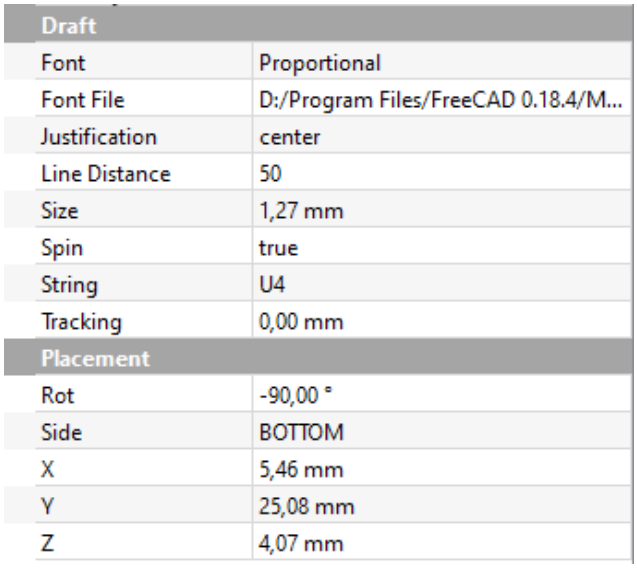
**Bottom Step Size**: distance between parts placed on bottom side of board

**Inverse**: switch exploded parts from top to bottom and conversely

**Top Step Size**: distance between parts placed on top side of board.

| Property | Value |
|---|---|
| **Base** | |
| Label | Explode |
| Active | true |
| Bottom Step Size | 10,00 |
| Inverse | false |
| Top Step Size | 10,00 |

## Annotation/Object Name/Object Value



| Draft | |
|---|---|
| Font | Proportional |
| Font File | D:/Program Files/FreeCAD 0.18.4/M... |
| Justification | center |
| Line Distance | 50 |
| Size | 1,27 mm |
| Spin | true |
| String | U4 |
| Tracking | 0,00 mm |
| **Placement** | |
| Rot | -90,00 ° |
| Side | BOTTOM |
| X | 5,46 mm |
| Y | 25,08 mm |
| Z | 4,07 mm |

**Font:** font name

**Font file**: it is possible to use own font

**Justification**: text position according to X, Y values

**Line distance:** distance between lines (in % according to fonf size)

**Size**: font size

**Spin**: if parameter set to True text will keep rotation, parameter works for angle value >= 90deg

**Text**: text displayed by annotation object

**Tracking**: distance between letters

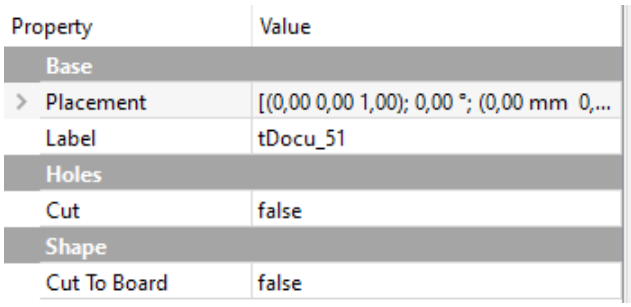
**Rot**: rotation value around Z axis

**Side**: text position on board (top/bottom side)

**X**: text position in X direction

**Y**: text position in Y direction

## Layer



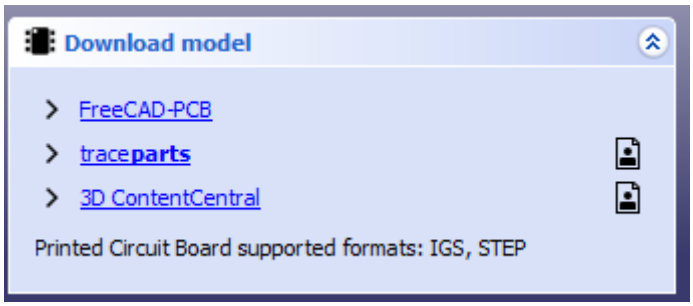
| Property | Value |
|---|---|
| **Base** | |
| Placement | [(0,00 0,00 1,00); 0,00 °; (0,00 mm 0,... |
| Label | tDocu_51 |
| **Holes** | |
| Cut | false |
| **Shape** | |
| Cut To Board | false |

**Cut**: show/blank holes

**Cut To Board**: cut shape to board outline

# 3D MODELS

Workbench comes without 3D model so it is necessery to dowload them separatelly. You can bring up the 'Download models' window from the PCB Settings toolbar. Mentioned window contains links to sites when you can find free 3D models.




Registration is necessary to download models

There is also possibility to search for concrete model. To do this just right click on missing model in specification tree and choose PCB model → Find model on-line

**Models from FreeCAD-PCB (github site) are directly connected with default database.db file.**

**To add/remove paths you need to open Preferences window. More info you can find in section Customizing workbench.**

**Workbench supports 3D models saved in one of the following formats: STP/IGS**

**The default path is set to the “/Parts" folder which is located in the main PCB Workbench folder.**

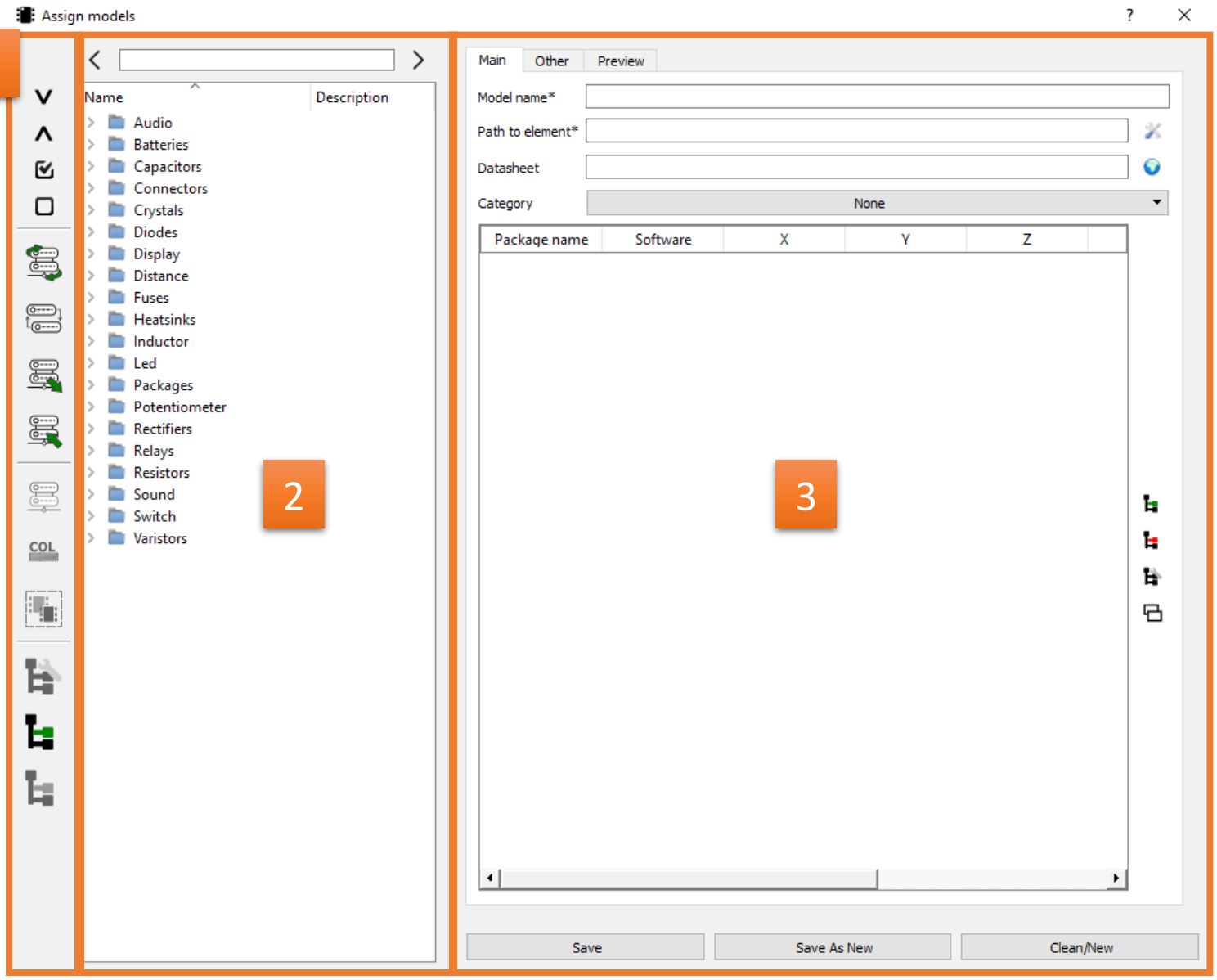
**It is recommended to keep parts outside PCB folder - to avoid data lost during workbench update.**

# ASSIGN MODELS

The ‘Assign models' window allow for assigning 3D models to corresponding part from one of the supported software.

It is necessery to assign 3D models and specific packages after downloading new components (not applicable for models downloaded from FreeCAD-PCB site).

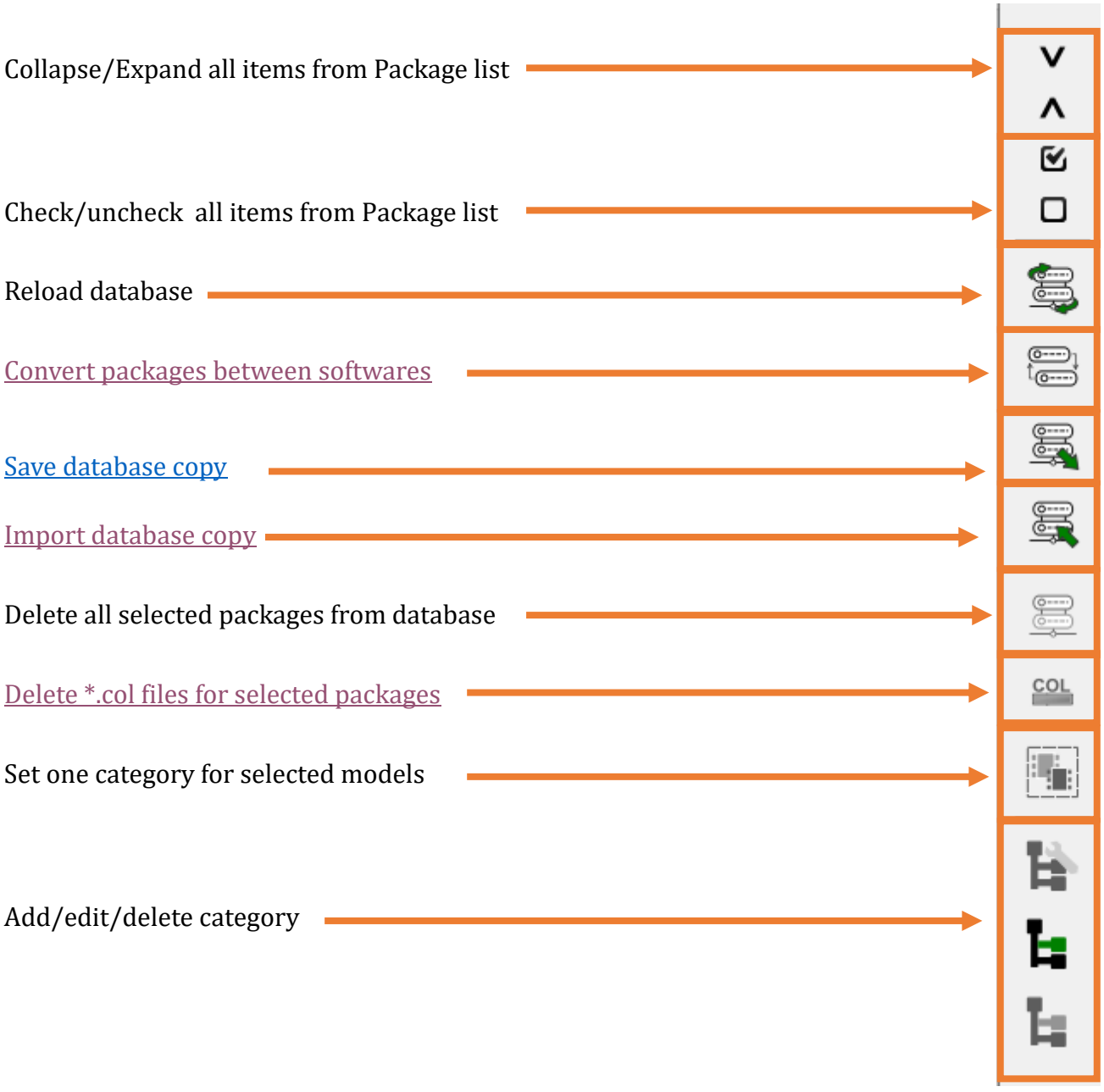
You can bring up the ‘Assign models’ window from the PCB Settings toolbar.



Window consists of three main areas:

1. Left column: contains functions necessary to manage parts in database
2. Middle column: contains list of all categories/packages saved in the database
3. Right column: allows to view/edit parameters for selected package

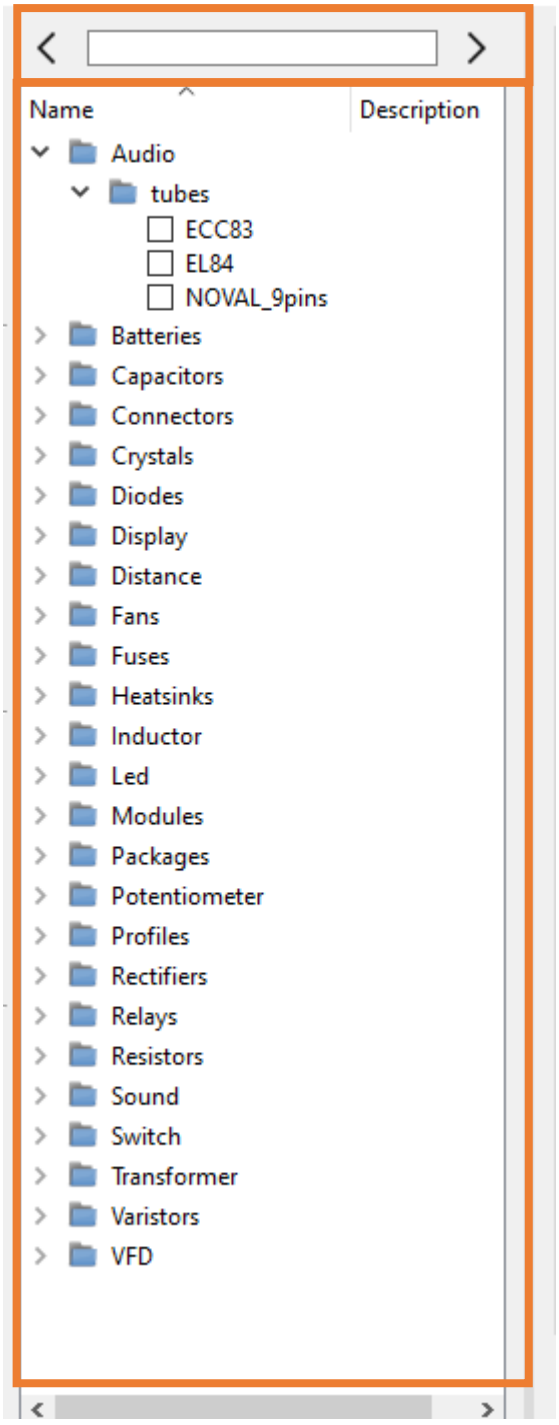
## Assign models – left column



## Assign models – middle column

Search block contains prev/next button and entry field for searching specified package

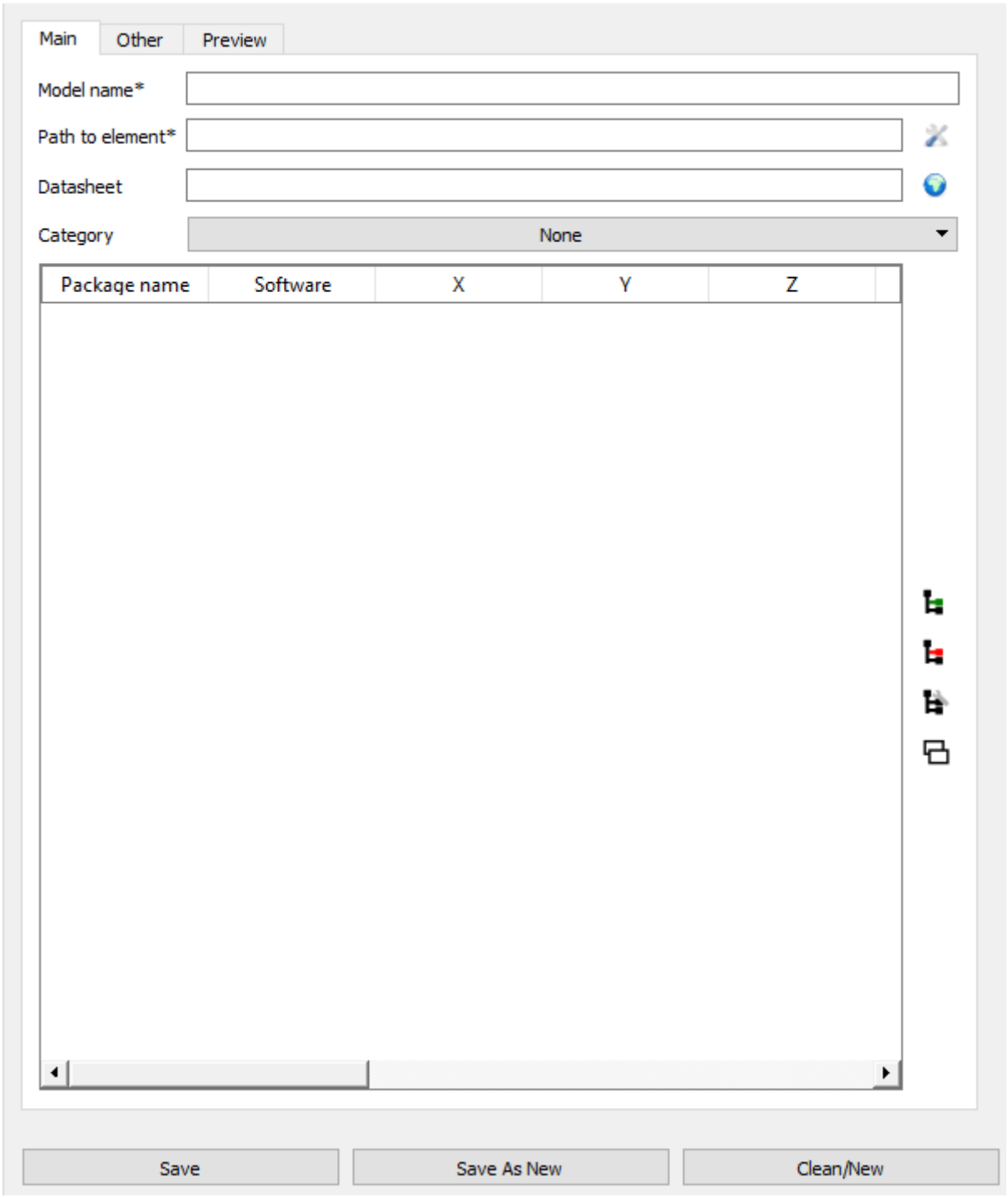
All models are grouped under categories (blue folder).



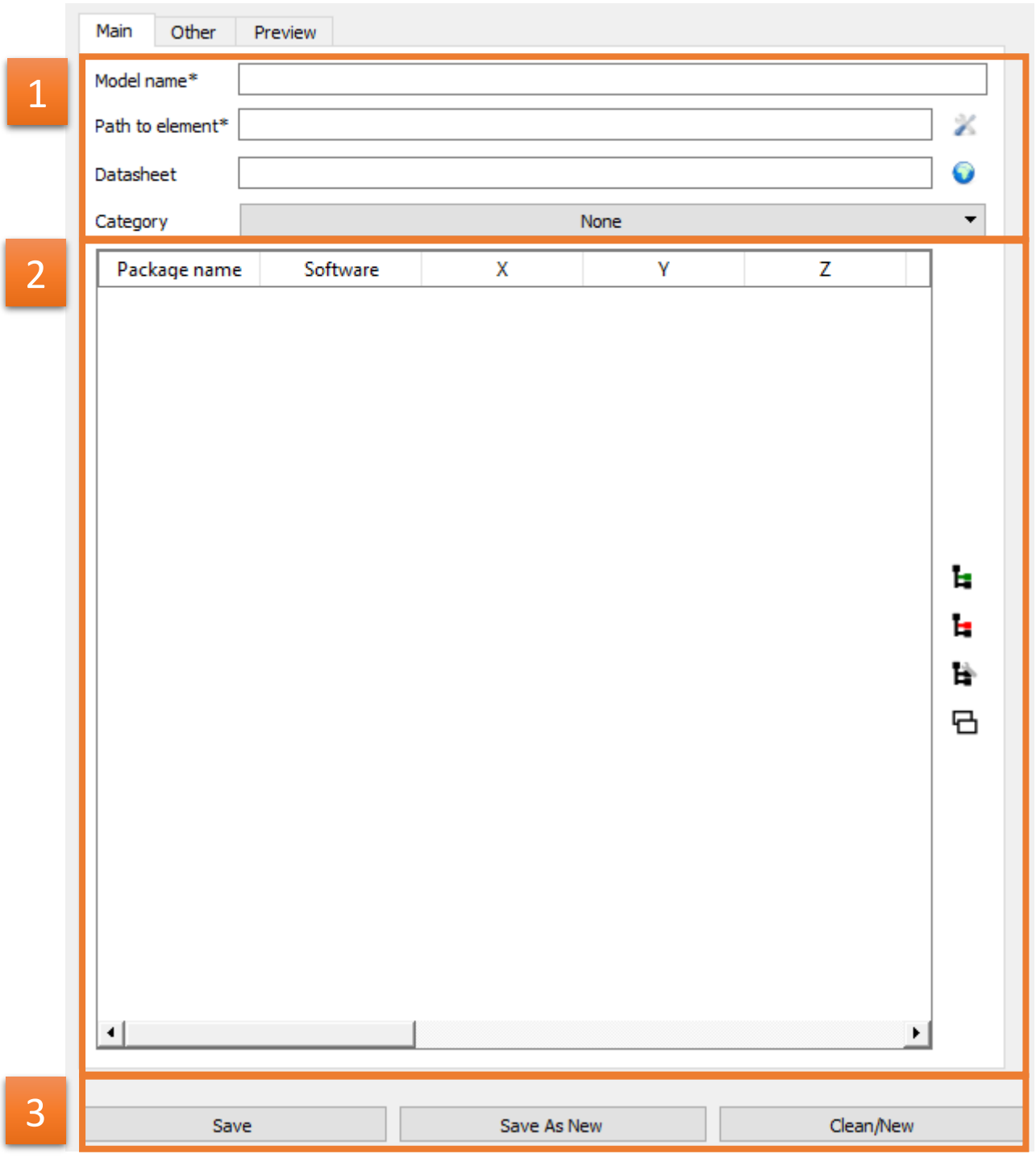
## Assign models – right column

Area is splitted to three main blocks:

- Main
- Other
- Preview

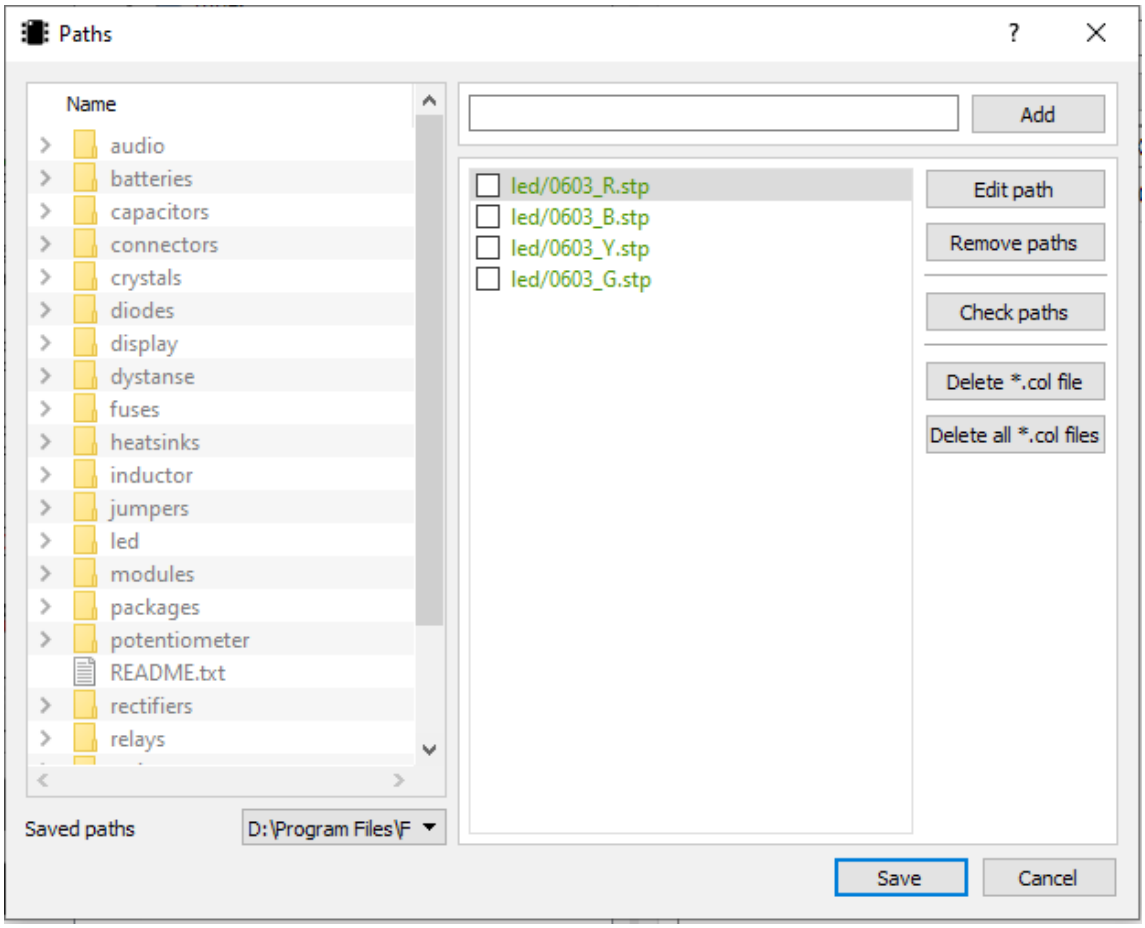


## Main tab


Main
Other
Preview
1
Model name*
Path to element*
Datasheet
Category
None
2
Package name
Software
X
Y
Z
3
Save
Save As New
Clean/New

1. Basic settings

- **Model name**: any name for model
- **Datasheet**: you can specific path to datasheet for package (url or path to pdf file)
- **Category**: define under what category model will be placed. To keep it without category select 'None'
- **Path to element**: path to assigned 3D models – it is available to assign more then one 3D representation for one model. To add/edit/delete path click button on the right – new window will appear




To add new 3D model just select it in the list on the left and click 'Add'.

Option 'Check paths' will check if previously picked paths still exist (green color → YES, red color → NO).

For more informations about *.col files check section '*.col files'.

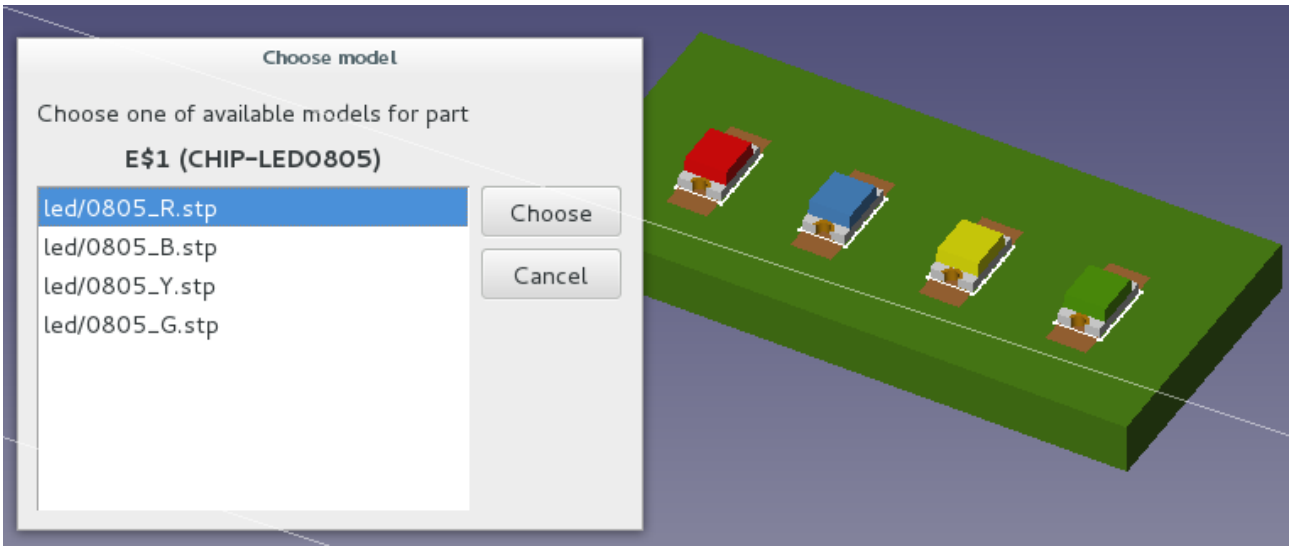
**If 3D model is under one of pre definied paths (in PCB Workbench preferences) you will see only relative path (no absolute).**

**If something will be not ok with model after loading board (for example no colors) or model will not automatically update despite the new 3D file use function 'Delete *.col file'.**

**Workbench supports 3D models saved in one of the following formats: STP/IGS**

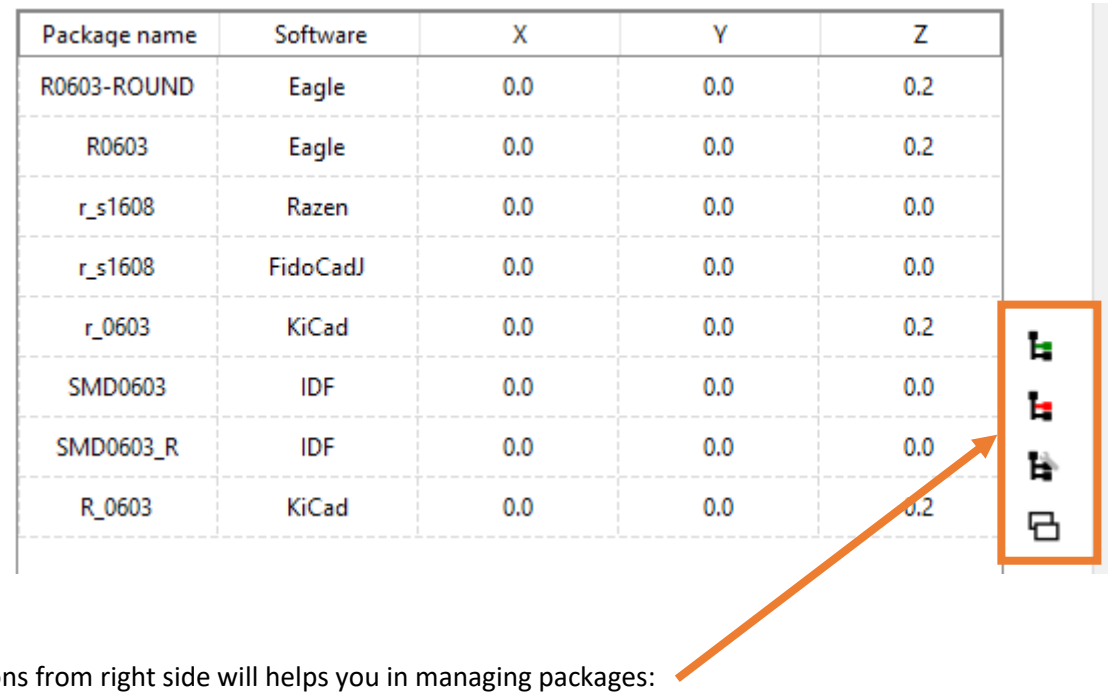
## Multi model definition for one part

This function is useful for parts which only different is color – the same correction values are set for all models. For packages where we set multi models, special window will appear during board loading or parts updating.




2. Models definitions

For each model you can define as many packages as you need (there are no limits).



| Package name | Software | X | Y | Z |
|---|---|---|---|---|
| R0603-ROUND | Eagle | 0.0 | 0.0 | 0.2 |
| R0603 | Eagle | 0.0 | 0.0 | 0.2 |
| r_s1608 | Razen | 0.0 | 0.0 | 0.0 |
| r_s1608 | FidoCadJ | 0.0 | 0.0 | 0.0 |
| r_0603 | KiCad | 0.0 | 0.0 | 0.2 |
| SMD0603 | IDF | 0.0 | 0.0 | 0.0 |
| SMD0603_R | IDF | 0.0 | 0.0 | 0.0 |
| R_0603 | KiCad | 0.0 | 0.0 | 0.2 |

Buttons from right side will helps you in managing packages:

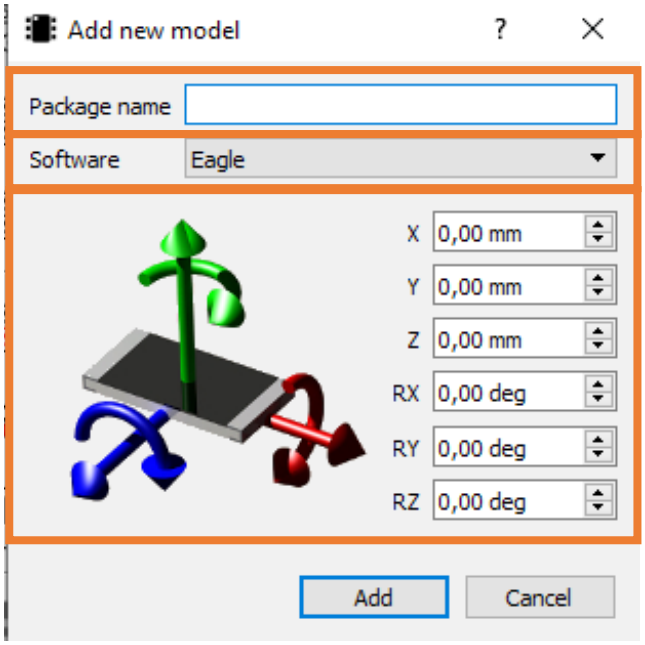
- 'Add' special window will appear, that allow you to set parameters for new package.
- 'Edit' button will appear window, that will contain all settings for current selected model.
- 'Delete' button will delete from database selected entry.
- Last button allows you to copy existing entry and save it in database under new name

## Adding new package

This field contains package type name taken from software used by you to create PCB boards.

From drop-down list You need to choose software name for with this entry will be connected.

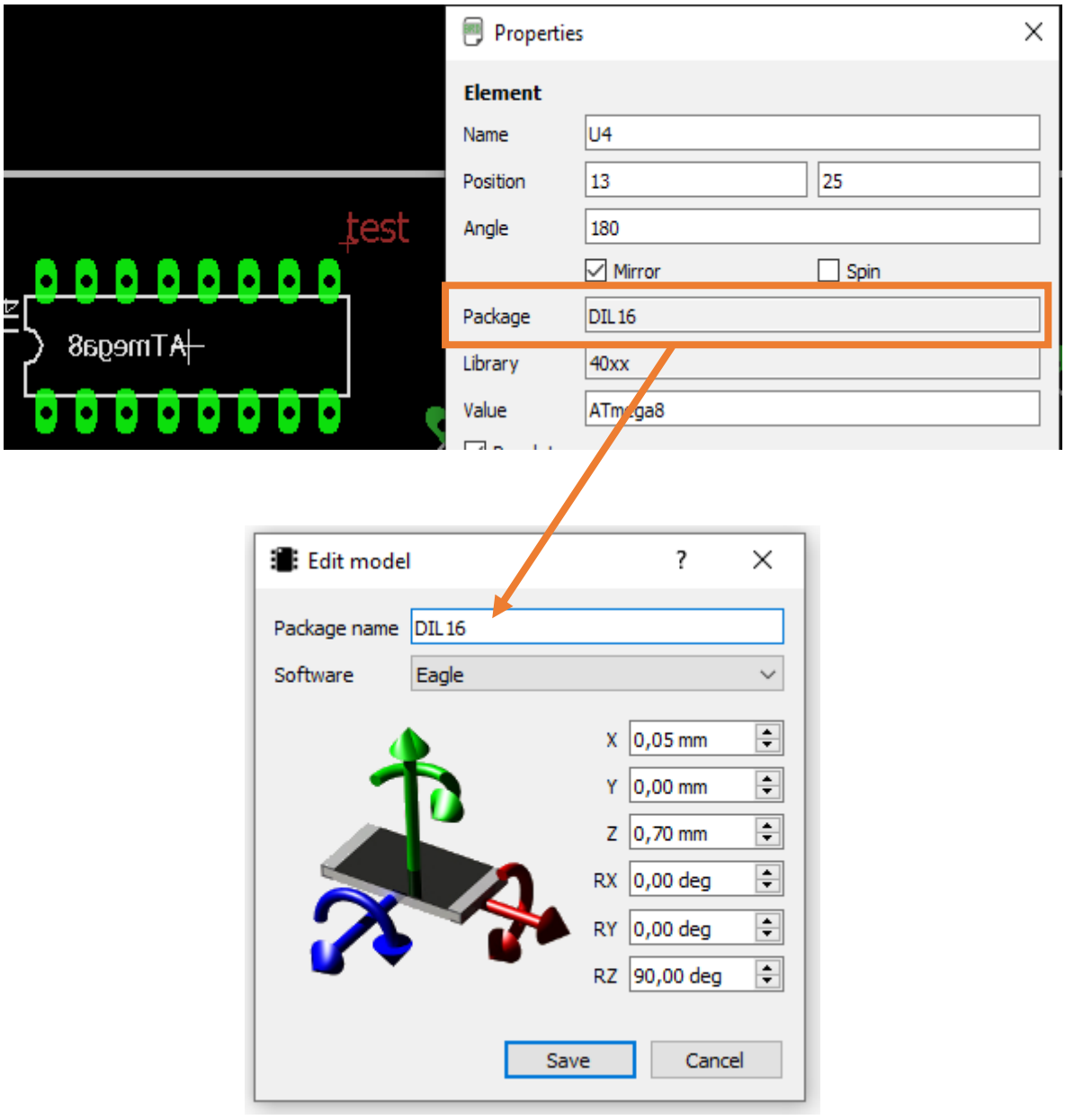
X , Y, Z, RX, RY, RZ parameters are correction values used to correctly placement 3D model




**Package name is closely related to the software which from PCB files comes**

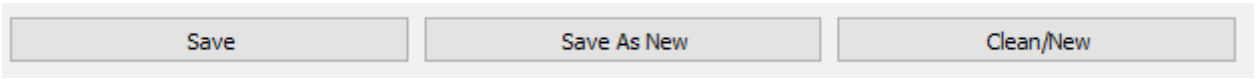
## Example: Defining a new package DIL16

Eagle: package name = DIL16




3. Buttons

To save specified model in database you need to use one of the available on the bottom buttons.



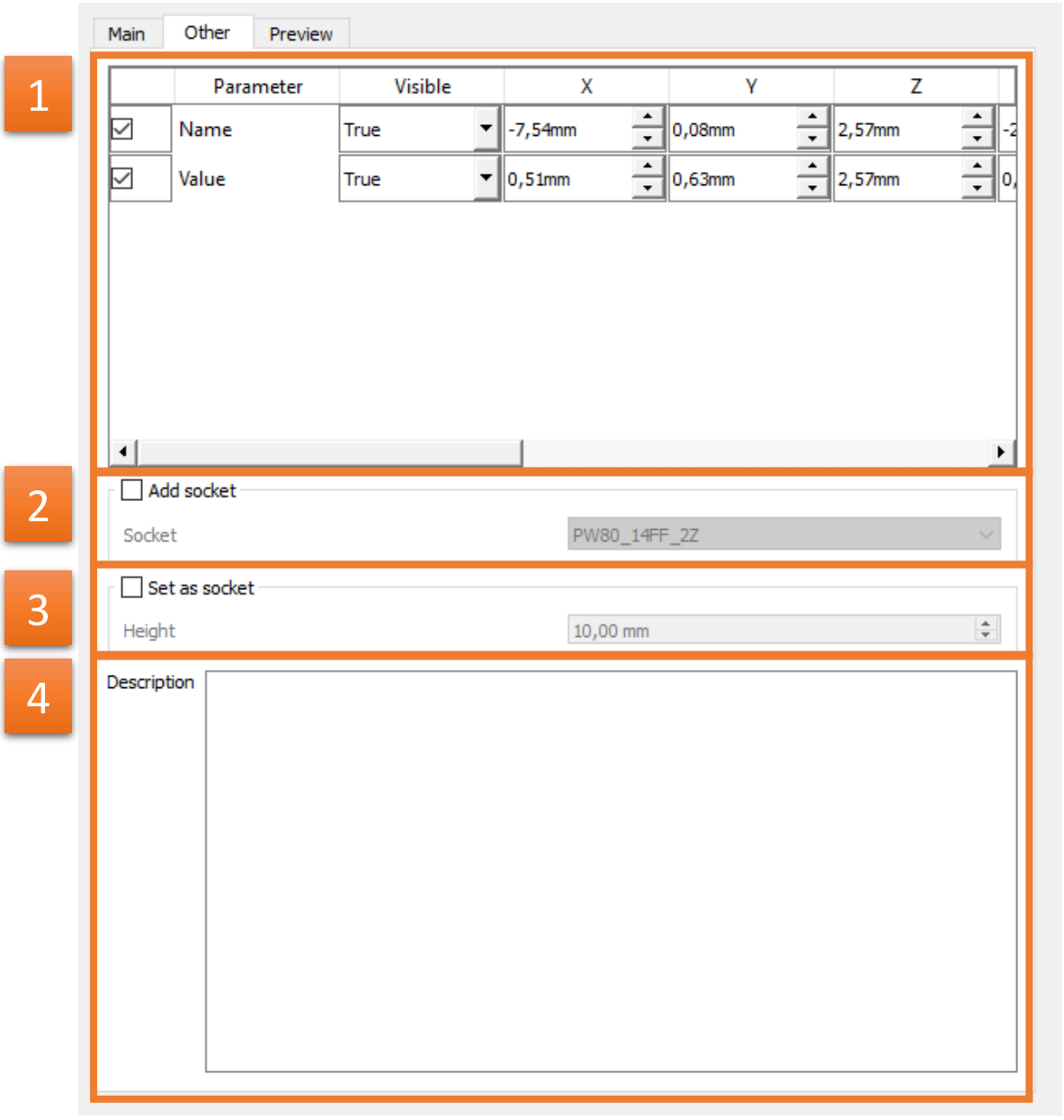

Save button will save form as new entry in database or will update existing model.

'Save as new' will save existing entry in database under new package name.

'Clean/New' button clean form.

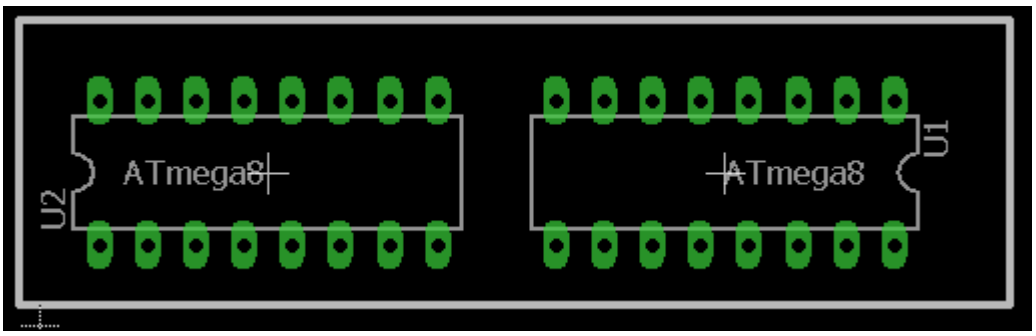
Close button will appear only for GNU/Linux users.

## Other tab

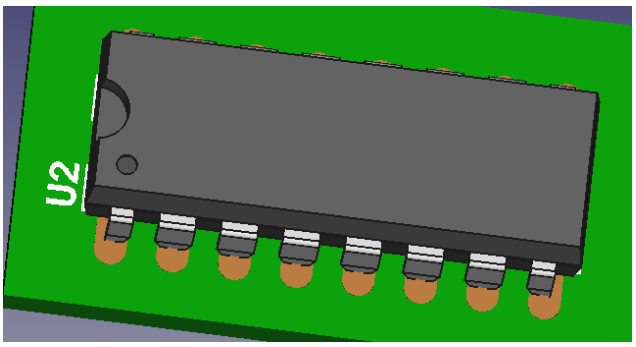


1. 'Adjust part name/value' **:** option allows to automatic placing objects name/value in specific position.

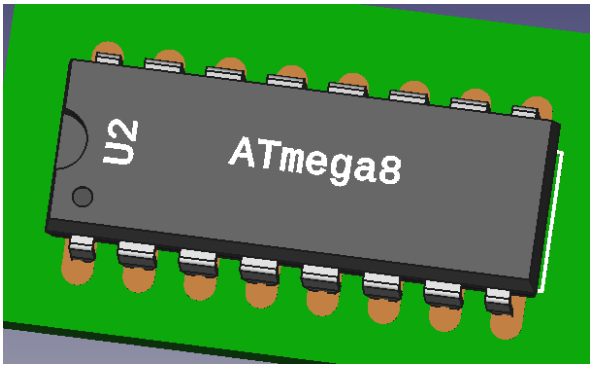
**Example for DIL16:**




Board created in external software.




'Adjust part name/value' = OFF




'Adjust part name/value' = ON

Board view after importing to FreeCAD.

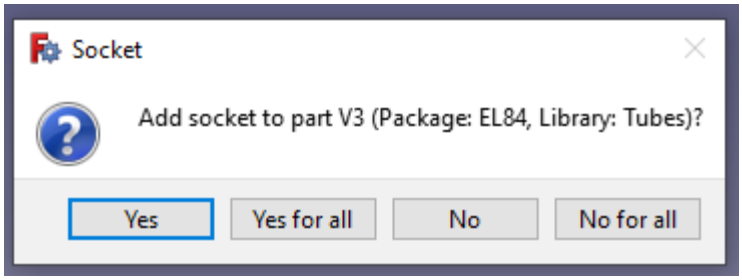
| | Parameter | Visible | X | Y | Z | RZ | |
|---|---|---|---|---|---|---|---|
| ☑ | Name | True | -7,54mm | 0,08mm | 2,57mm | -270,00deg | 1 |
| ☑ | Value | True | 0,51mm | 0,63mm | 2,57mm | 0,00deg | 1 |

Parameter activation

## 2. Set socket for model

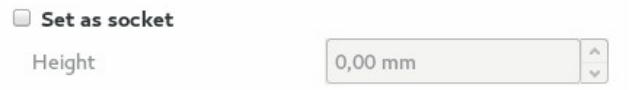
To add socket for model just mark checkbox for 'Add socket' and from drop down list choose socket 3D model name. In drop down list you will find only models marked before as sockets


For model where socket was specified special window will appear.




## 3. Set model as socket

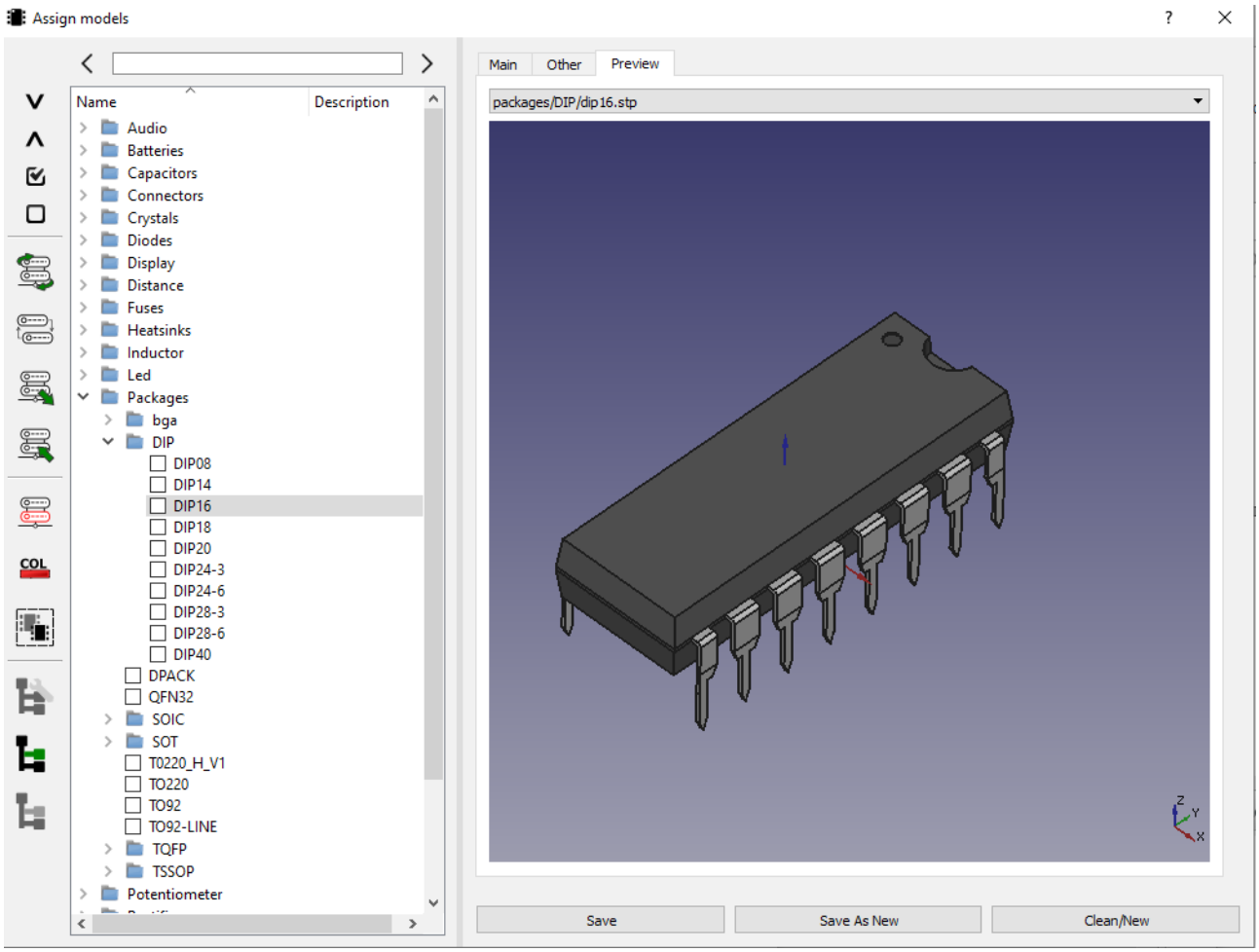
To set model as socket just mark checkbox for 'Set as socket' sign. Enter the height of the socket in the spinbox.




## 4. Description

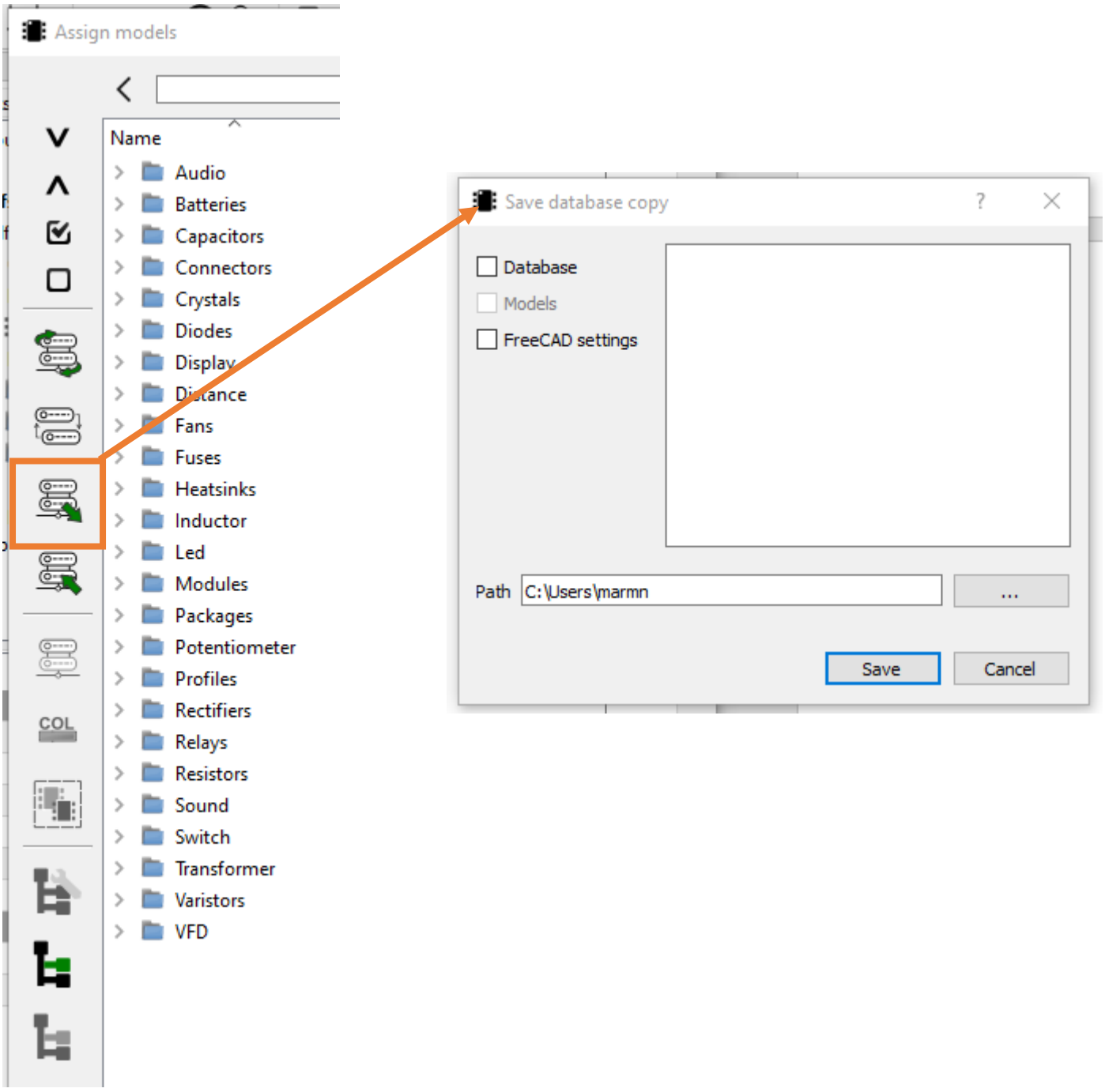
## Preview tab

It is possible to see 3D model in last tab - ‘Preview’.



# SAVE DATABASE COPY

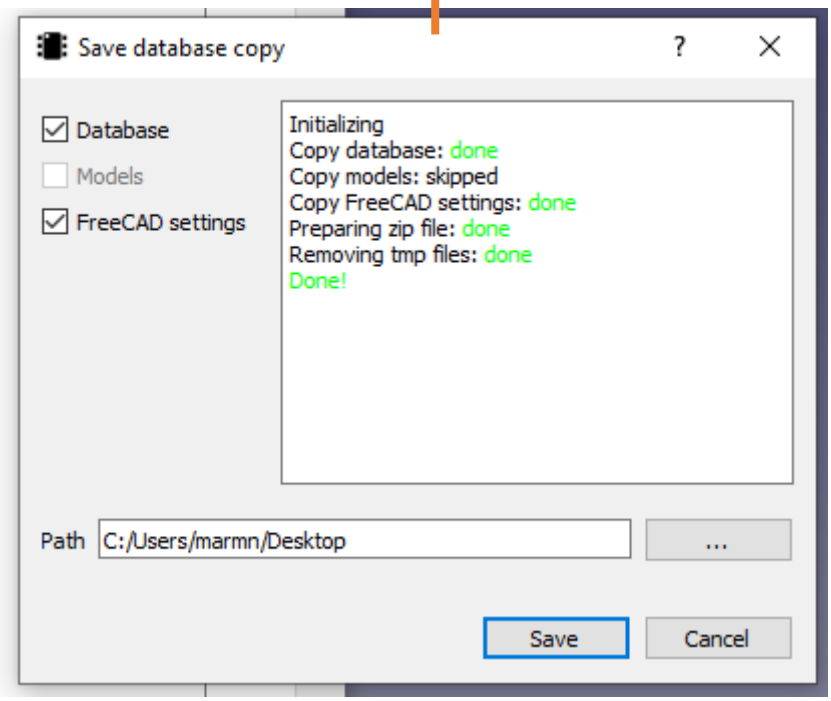
It is possible to easily prepare copy of the database and all settings stored in FreeCAD. This option is only availably in ‘Assing models’ window.



**At the moment it is only possible to export database file and settings stored in FreeCAD.**

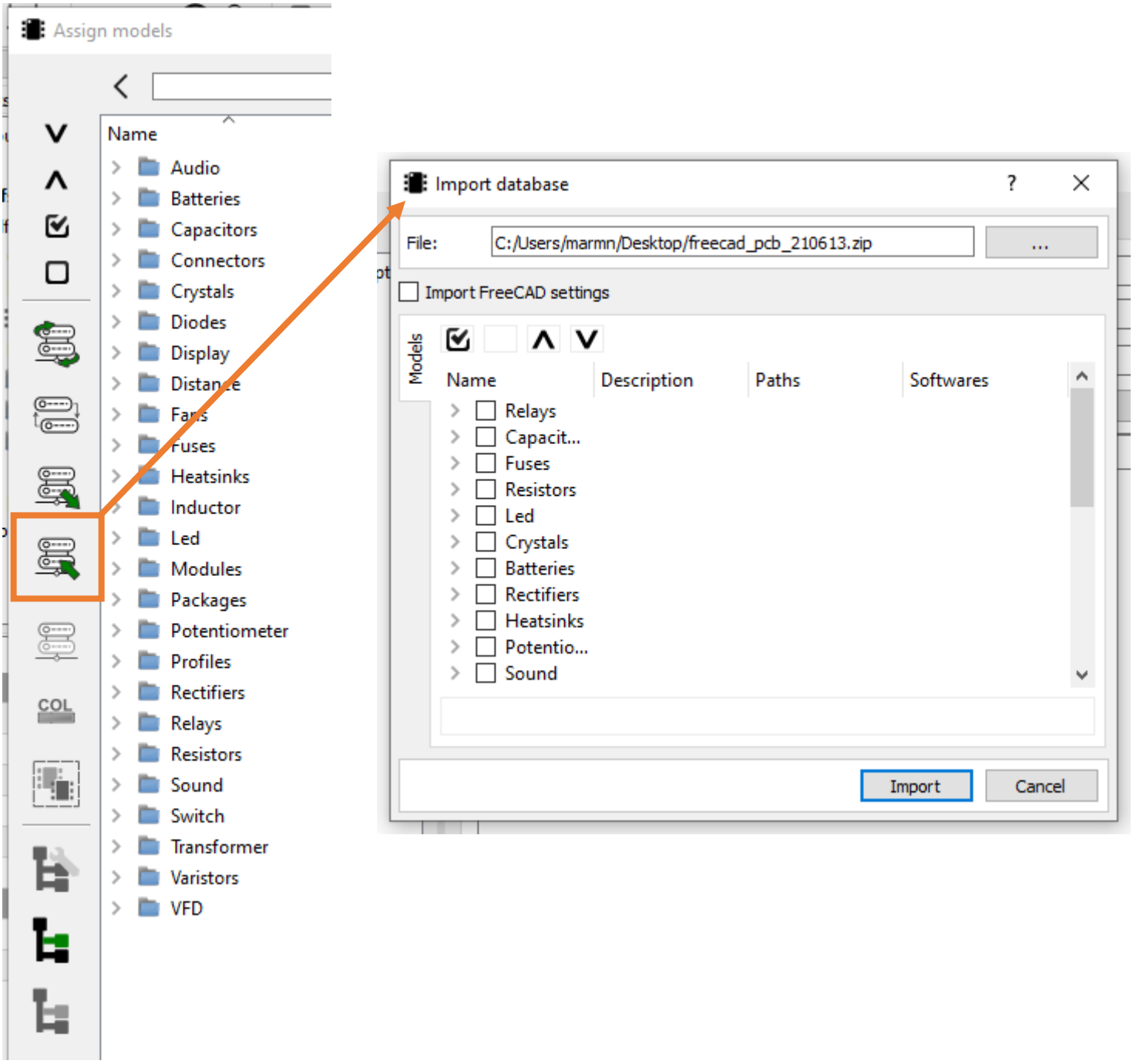
The effect of using this function is a * .zip file.

*freecad_pcb_210613.zip*

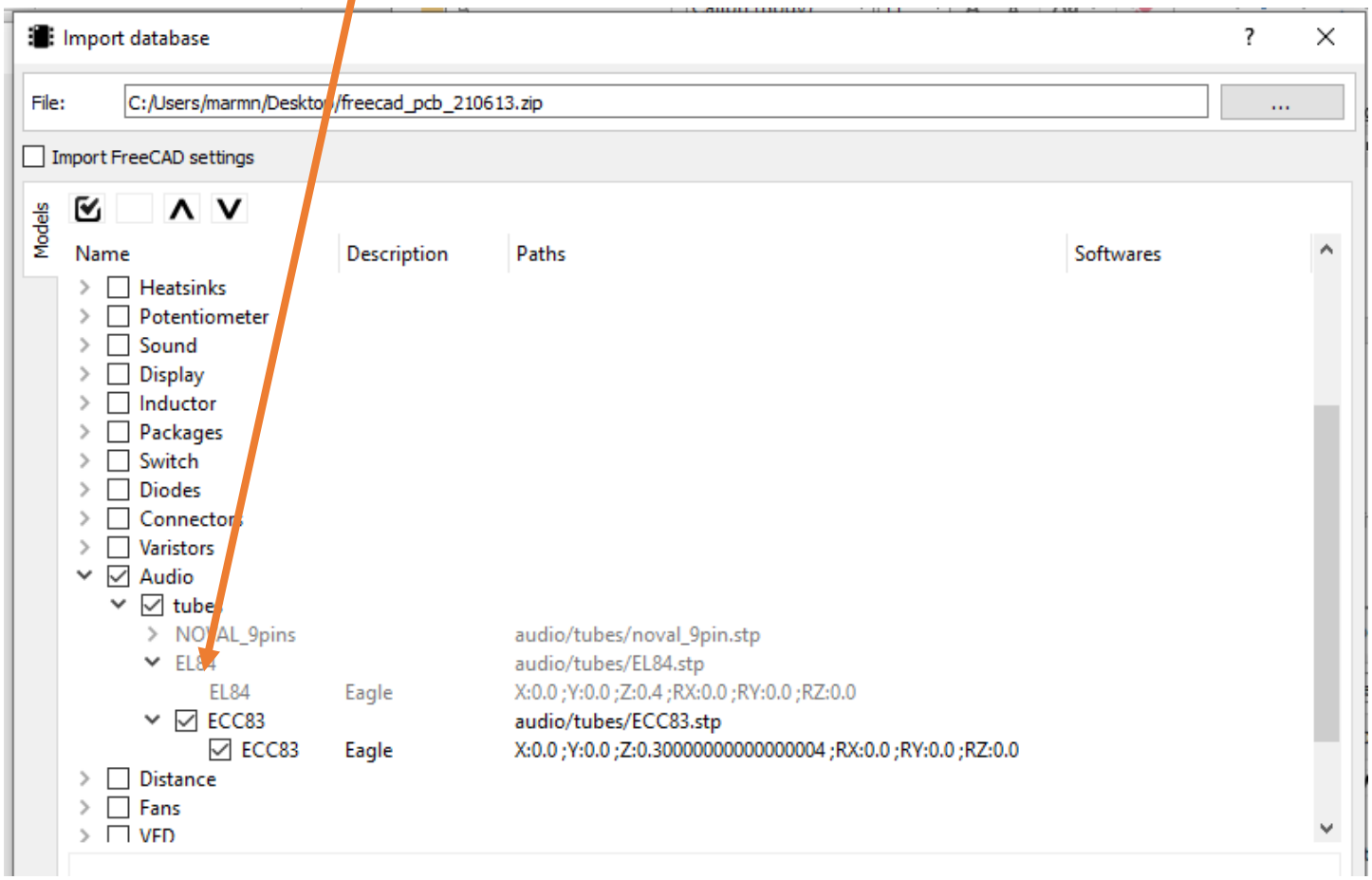


# IMPORT DATABASE COPY

To import previously saved copy you need to click button ‘Import database’, which is availably in ‘Assing models’ window.

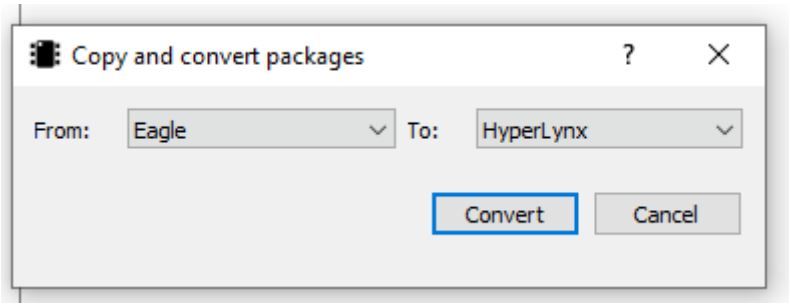


To import specific model just check it. Function will automatically compare models from imported file and saved in the existing database, which makes it impossible to import already existing models.



# COPY AND CONVERT PACKAGES

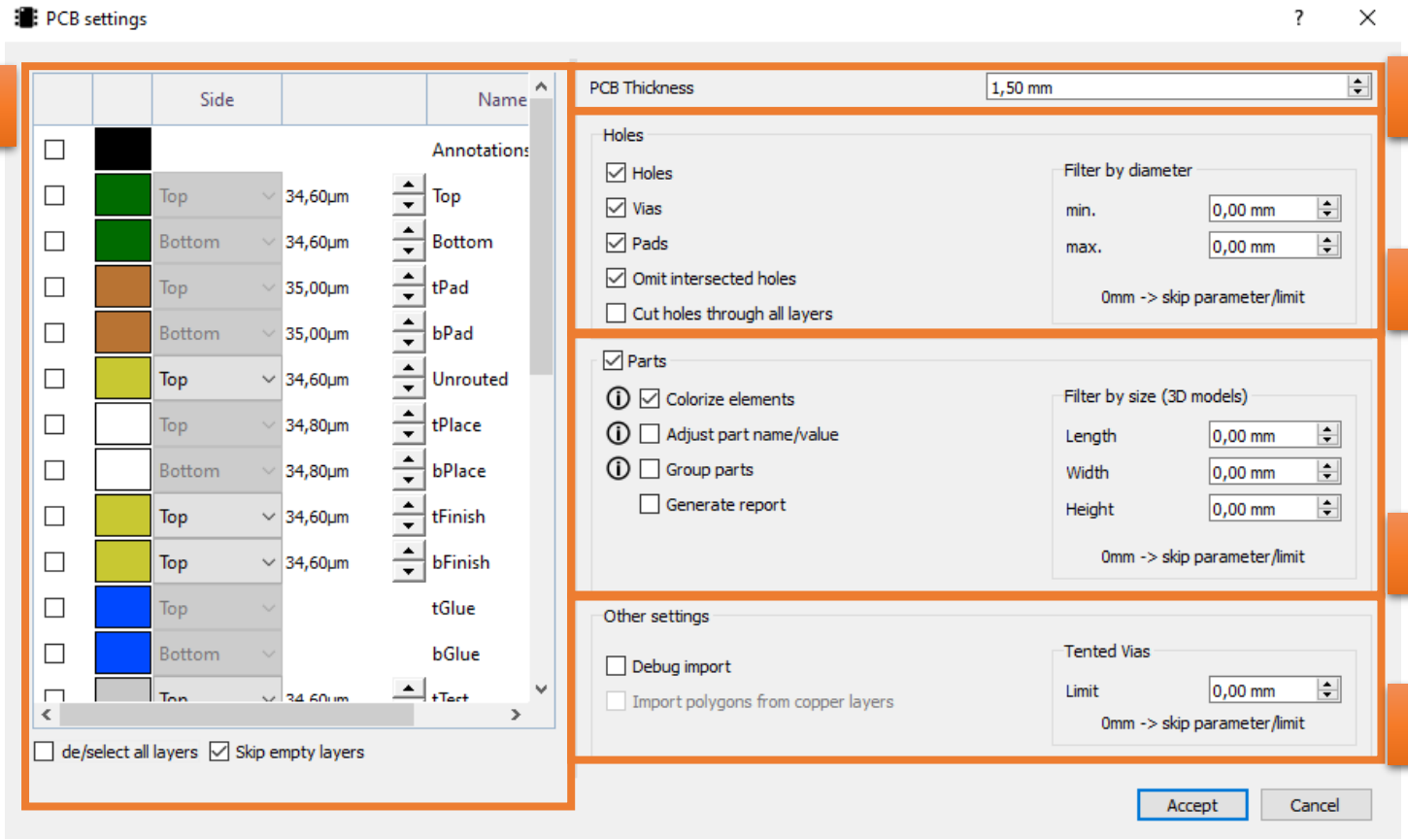
This function will allow you to easily and quickly to convert defined packages from one supported softwares to another.



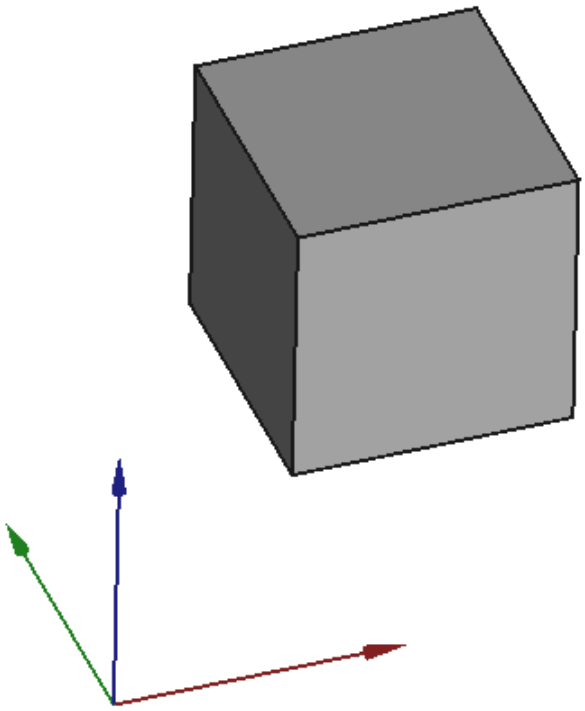
# WORKING WITH WORKBENCH

# OPENING/IMPORTING BOARD

During the opening / importing process, a special window will appear in which you can set the basic parameters of the board.




1. In first section You can choose, which layers will be loaded. Available layers depends from loading file type. Layer name and color are editable.
2. This section allow You to set PCB thickness. If file contain board thickness this value will be displayed in this field. Default value is 1.5[mm].
3. Third section contain basic settings about importing holes. Here You can decide what type of holes You want to import (hole/vias/pads) and set imported holes diameter range (min/max). Both parameter can be set separately.
4. Fourth area contains basic settings about importing parts. Here You can decide if You want to import parts, decide if they should contain colors, etc. Fields L/W/H allow You to decide about minimum length/width/height of 3D models which will be imported. All three parameter can be set separately.



5. Other settings

## Unit system

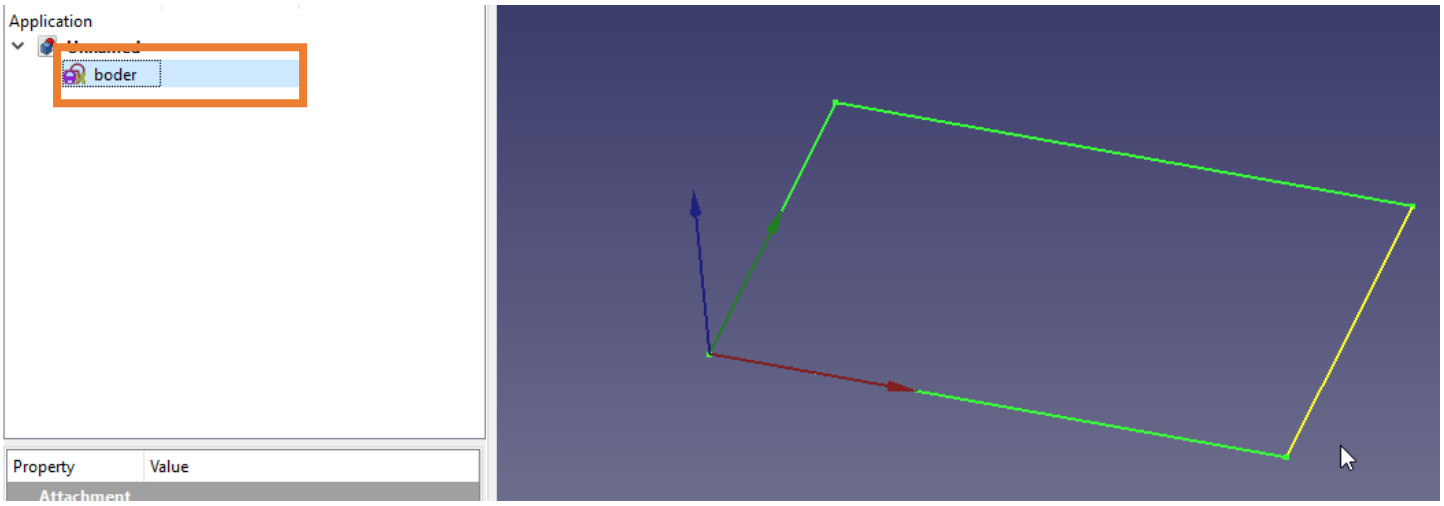
During board loading process units are changed to millimeters [mm].

**To skip a specific filter just set it to 0.**

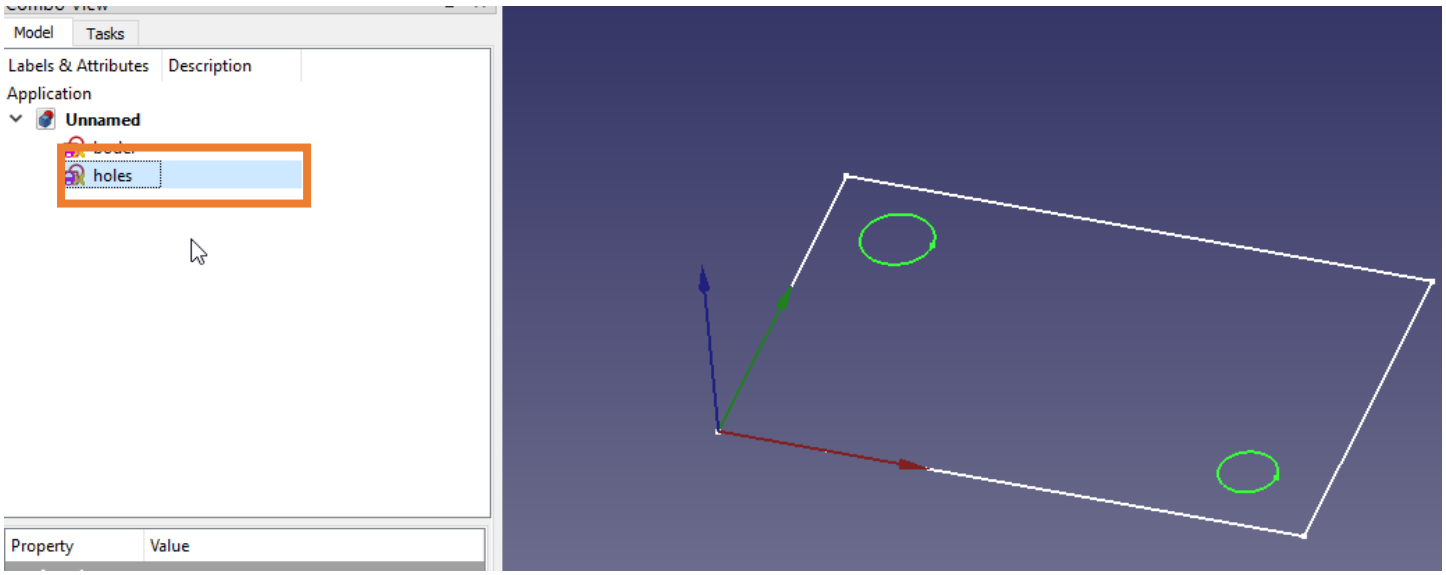
# CREATING BOARD FROM SCRATCH

In this section you will find information how to create a board and design its geometry from scratch.

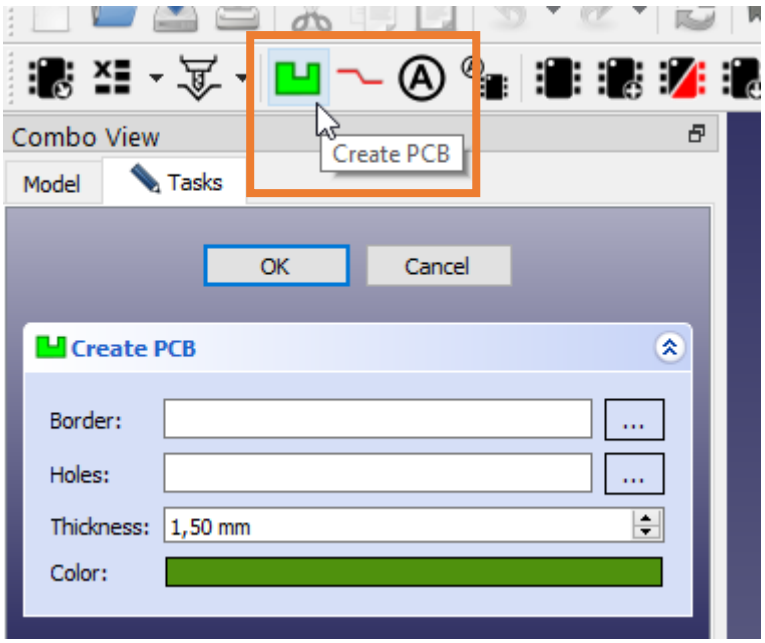
1. Create Sketcher with contour of the board. Sketcher name is not important



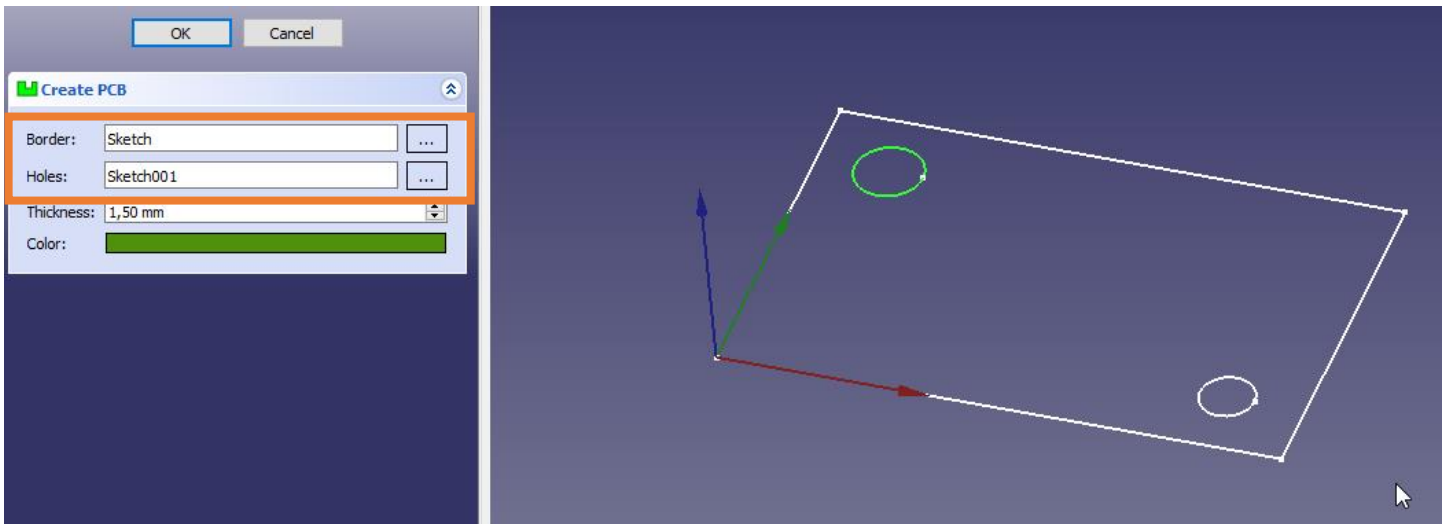

2. Create new Sketcher with holes. Sketcher name is not important



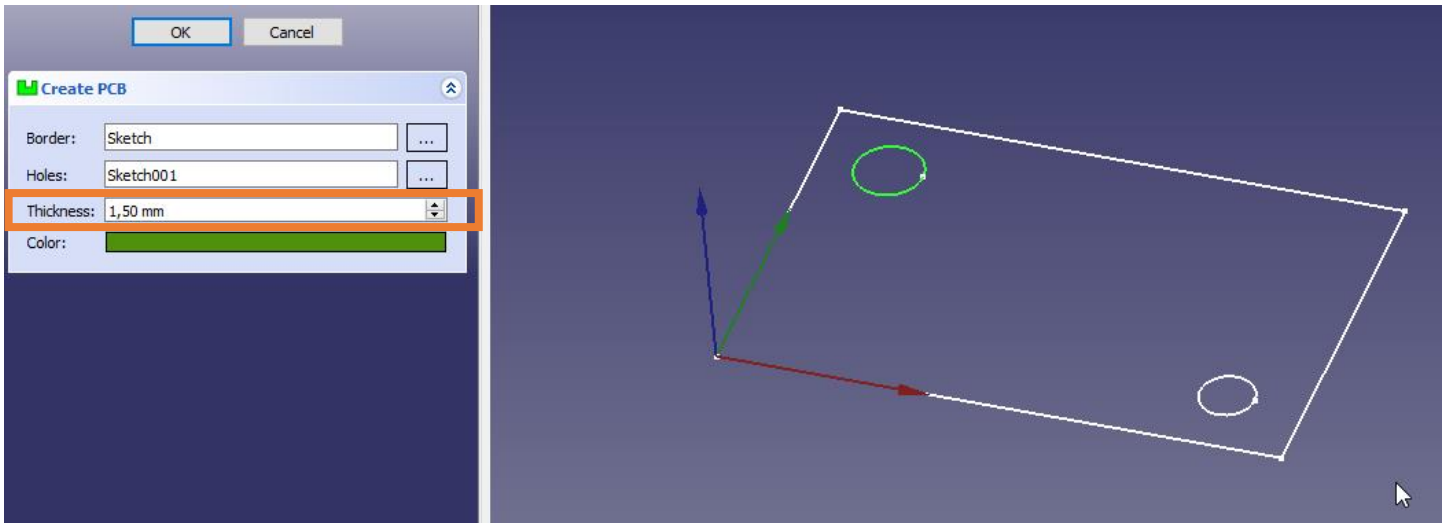
3. Click the **Create PCB** buton



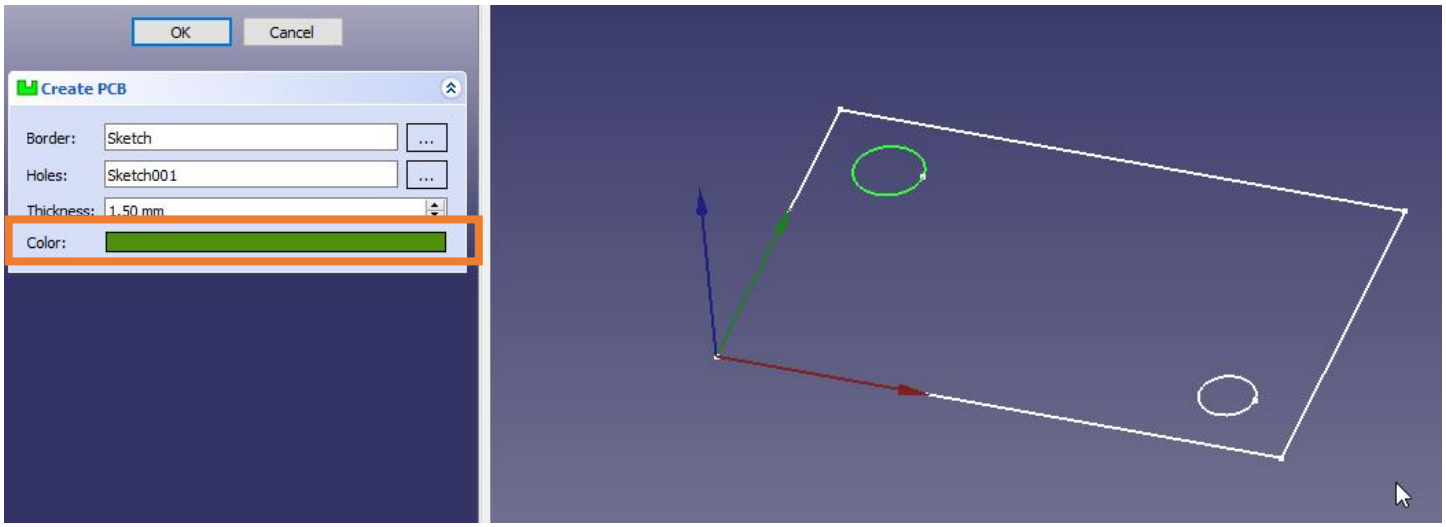

4. As Border set the Sketcher which contains contours of the board. Do the same with a Sketcher that contains holes



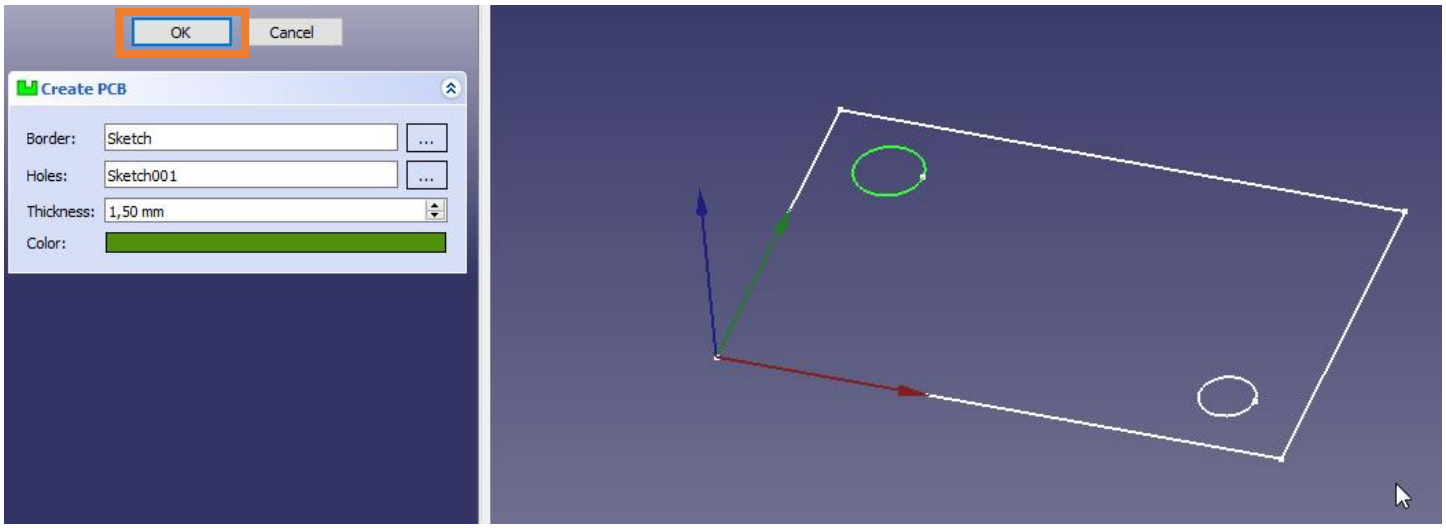
5. Set board **Thickness**



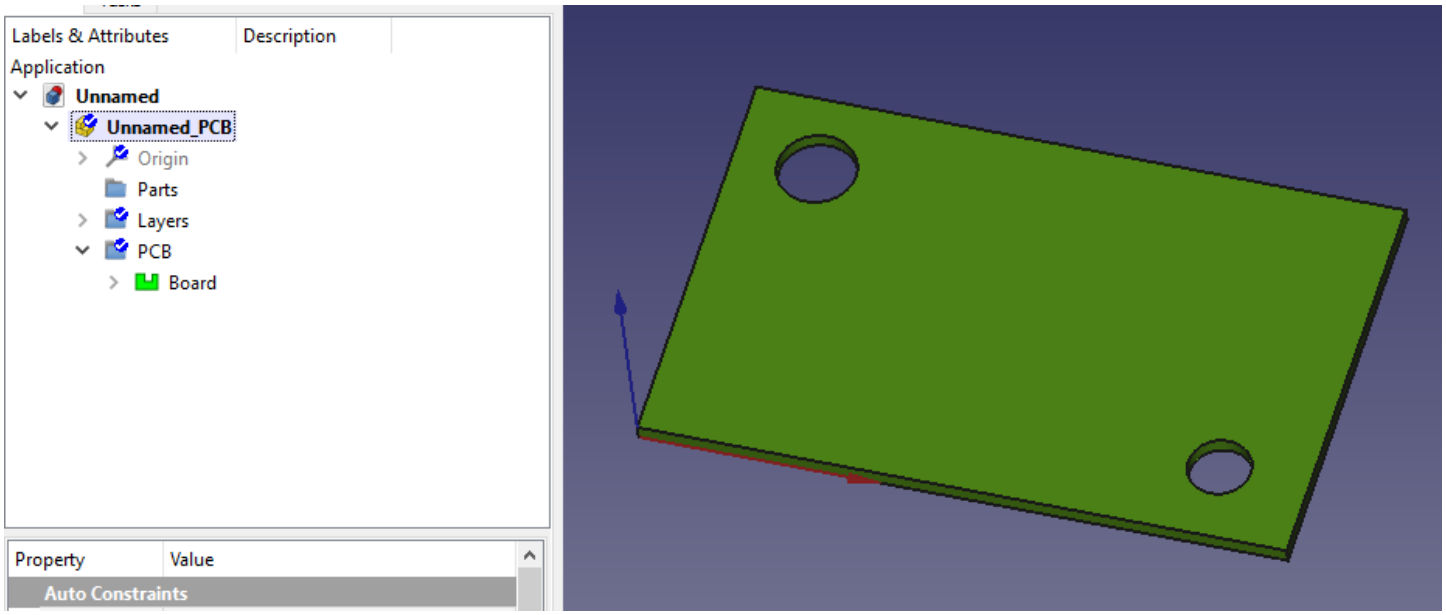

6. Set board **Color**



7. Click **OK** button to finish




The board should be generated according to the specified settings.



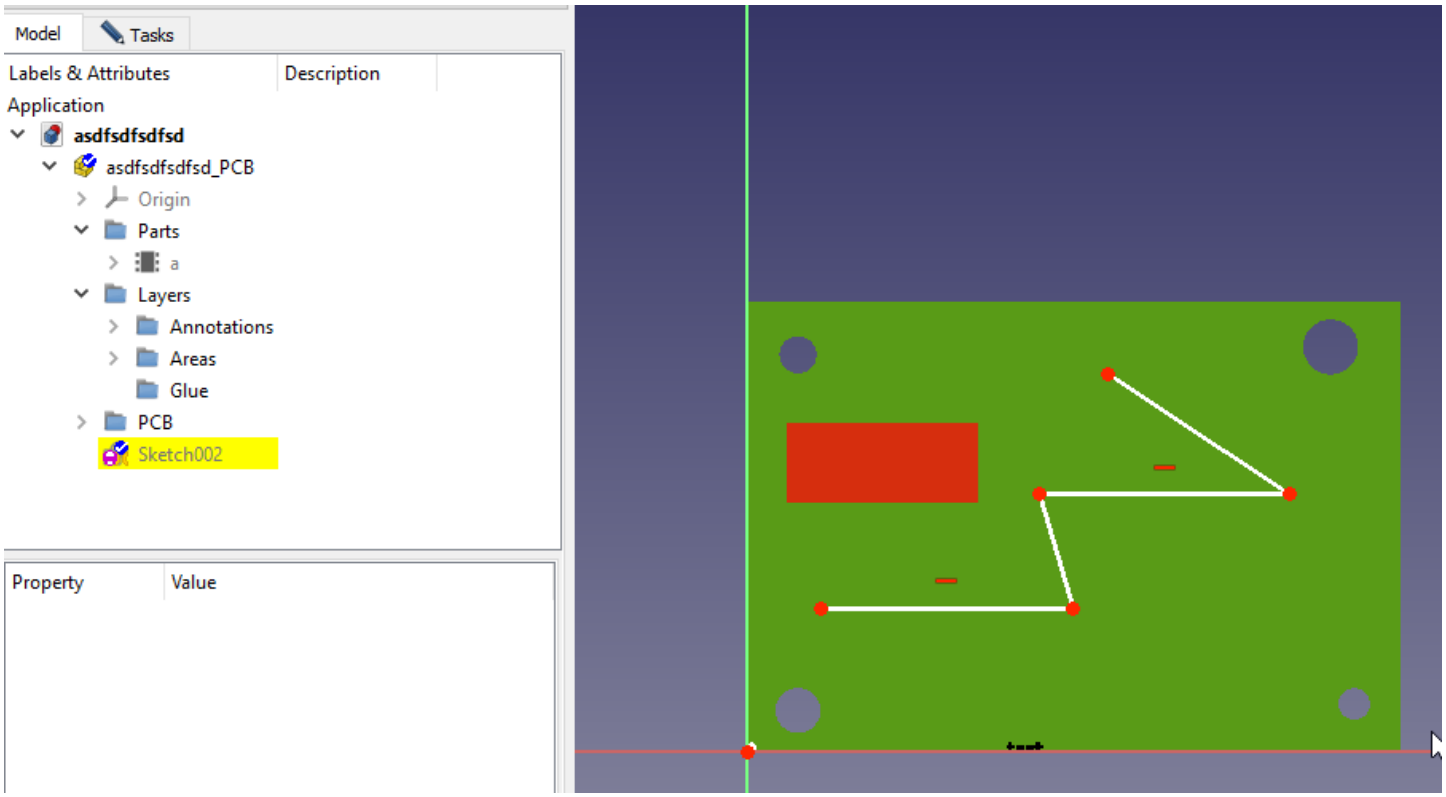
**Only one board can be generated per project**

**Even if there will be no holes in board, proper Sketcher need to be done**

# CREATING GLUE PATHS

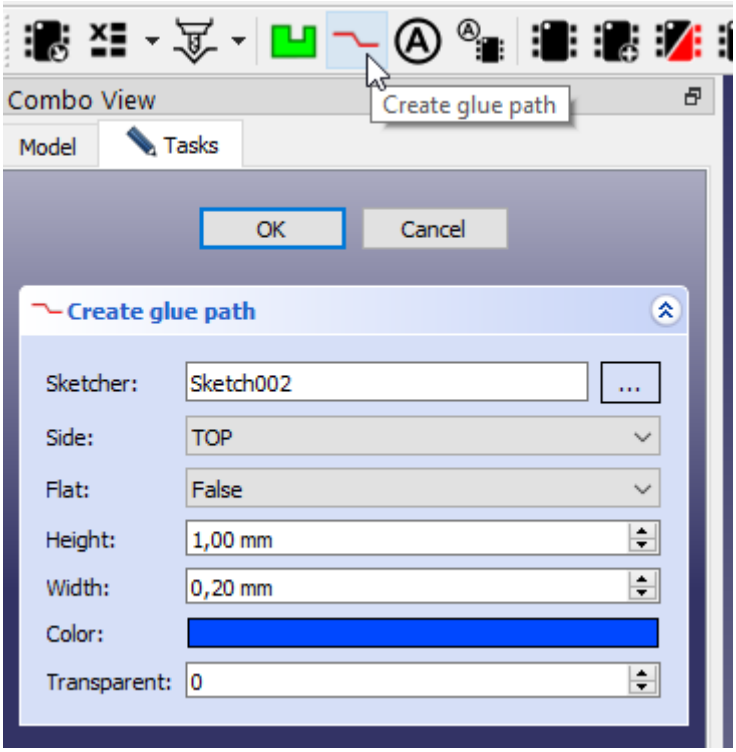
In this section you will find informations how to create glue path.

1. Create Sketcher with contour of the constraint area

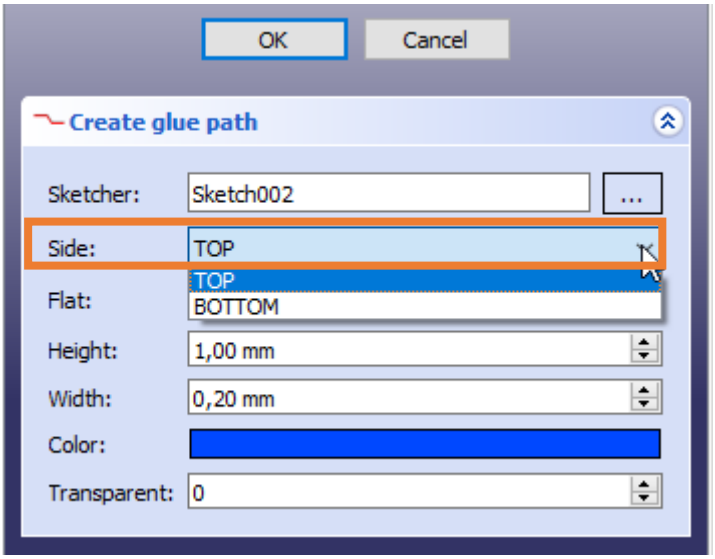



**Created seam does not have to create a closed shape**

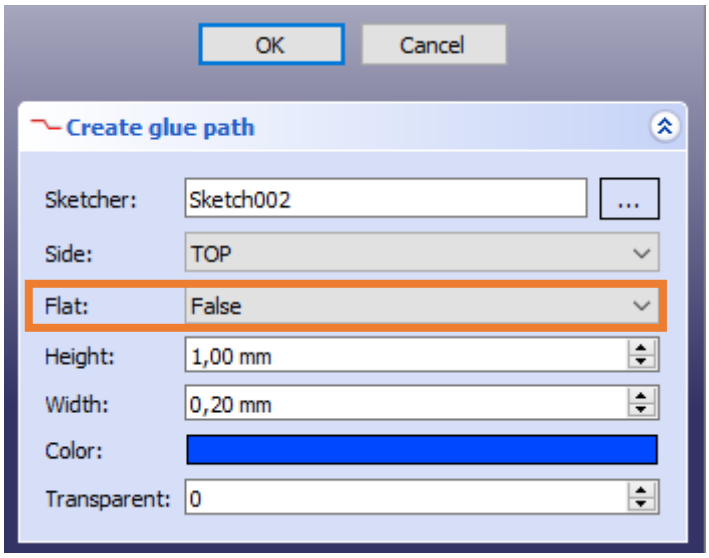
2. Select just created sketcher and click ‘Create glue path’ button



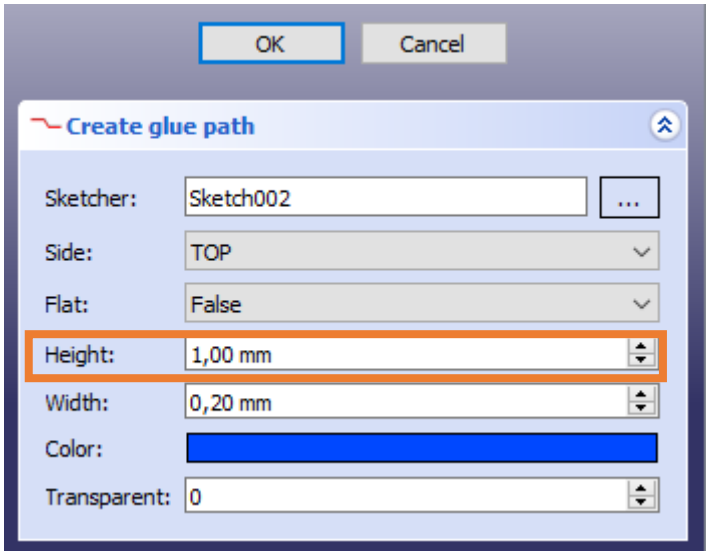

3. Set which side of the glue will be applied



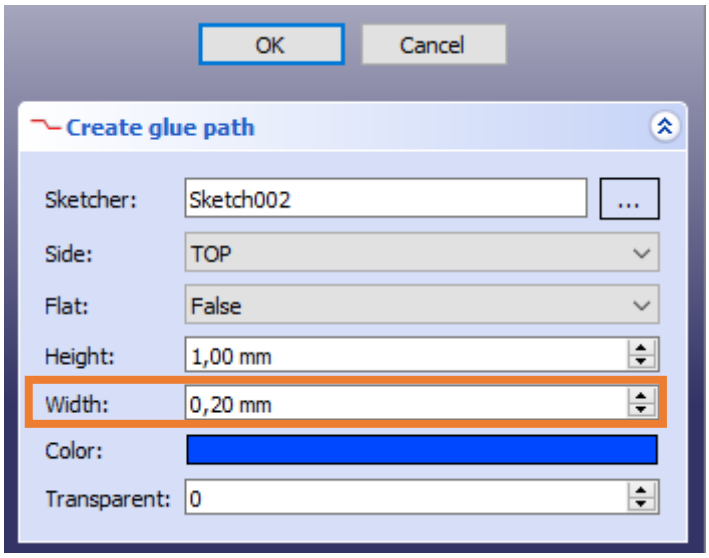
4. The Flat option determines whether the adhesive will be represented as a flat surface or as a 3D mode



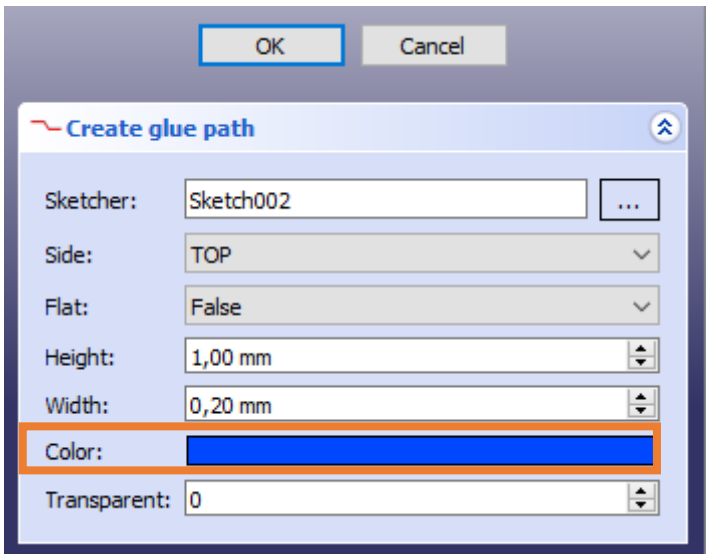

5. Height of the glue seam



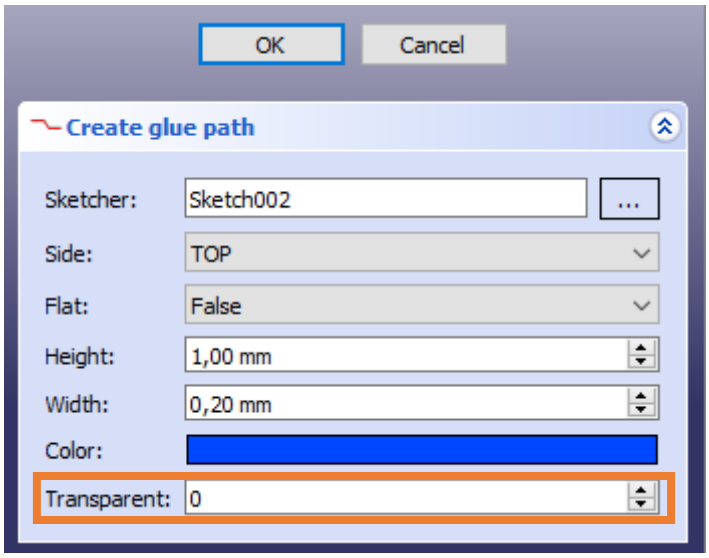
6. Width of the glue seam



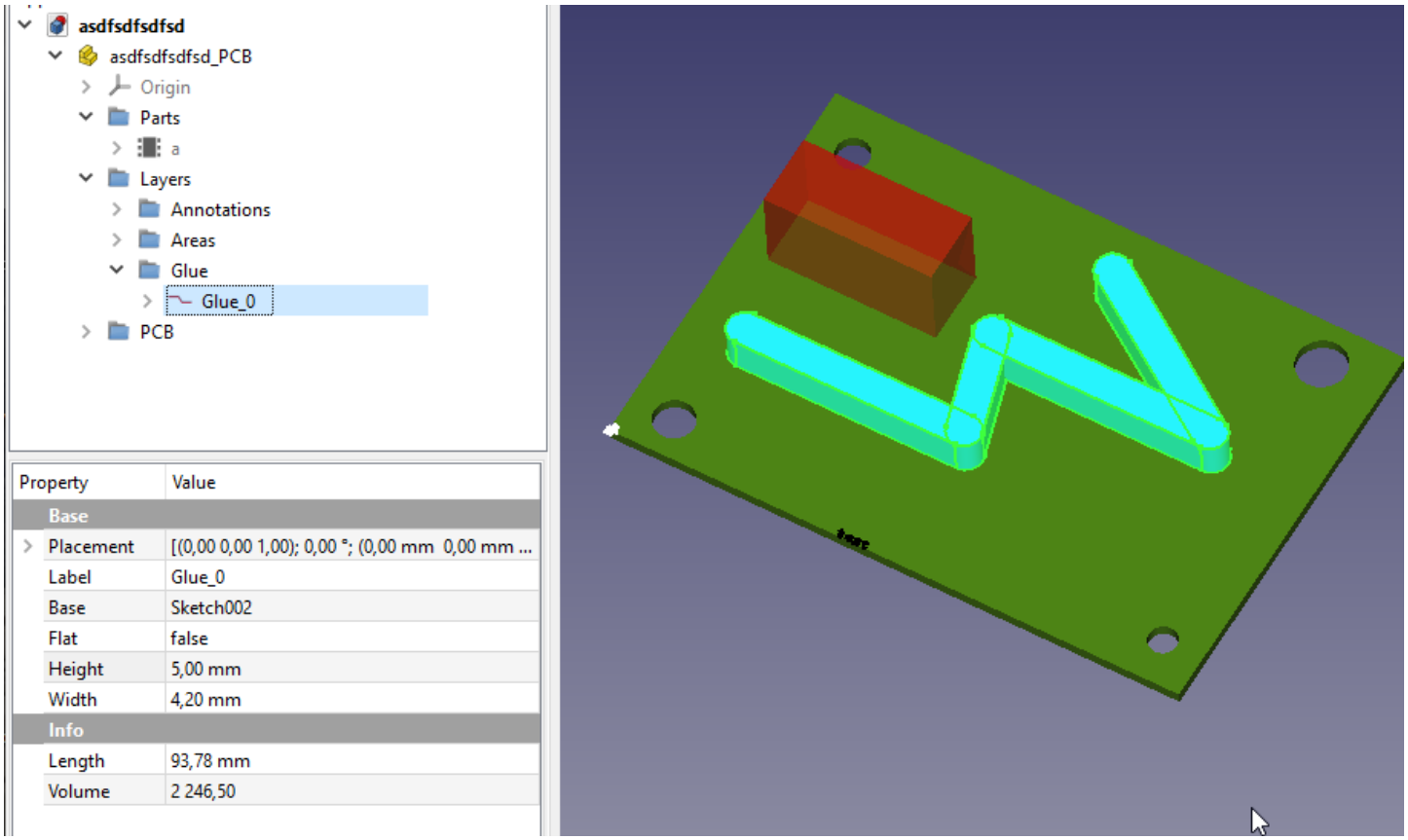

7. Glue seam color



8. Glue seam transparent




9. Click **OK** button to finish



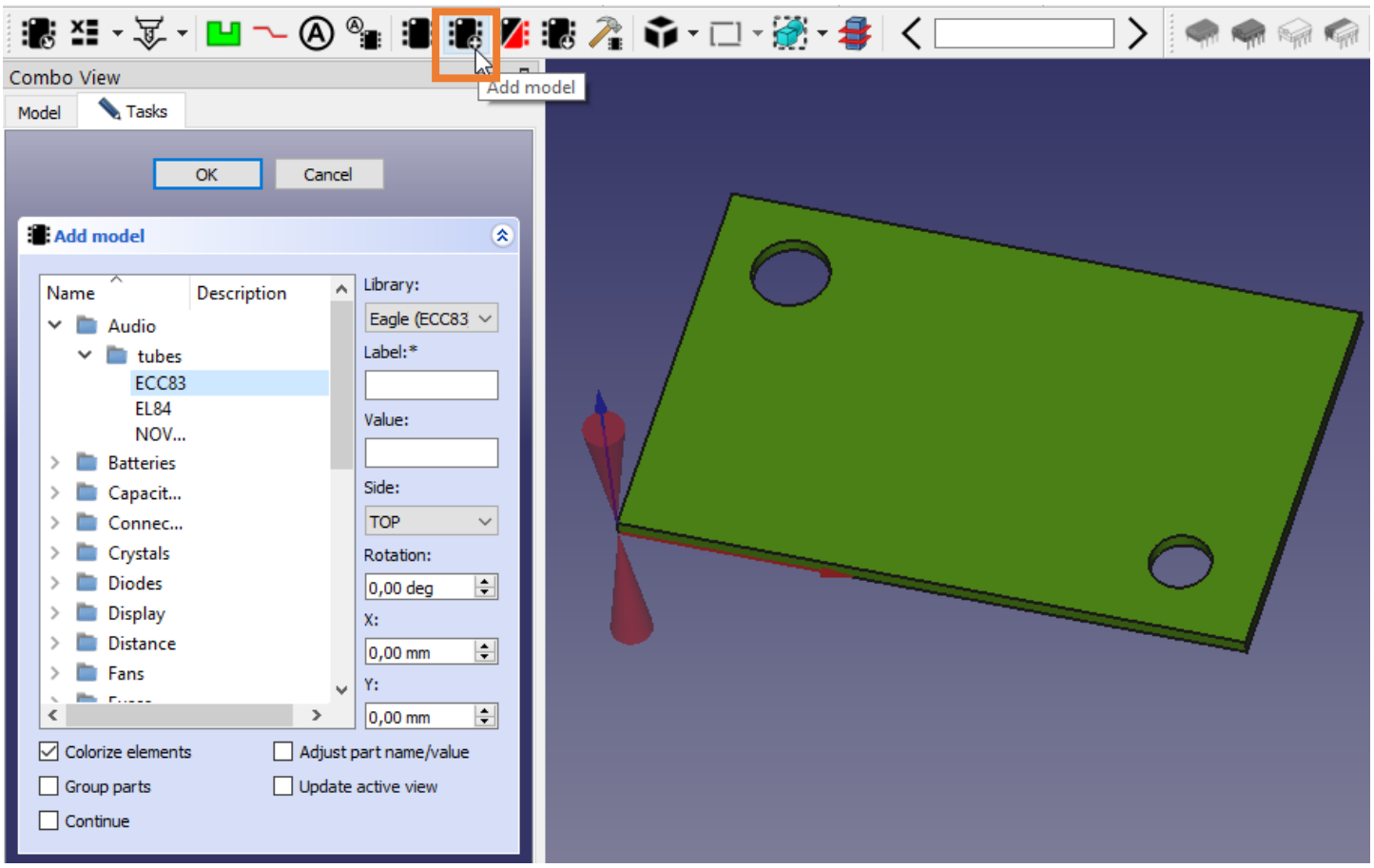
# ADDING ANNOTATIONS

In this section you will find informations how to add annotations to project.

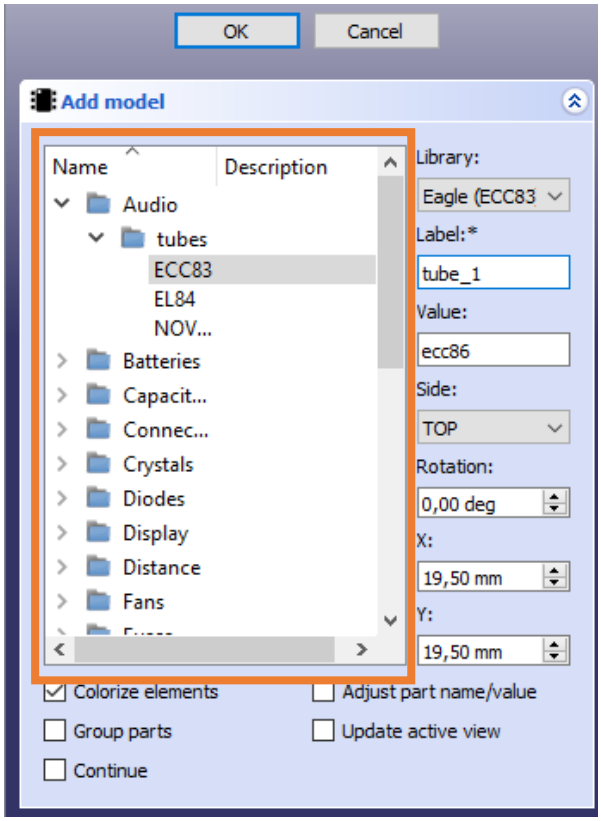
# ADDING NEW MODELS

In this section you will find informations how to add new component to existing board.

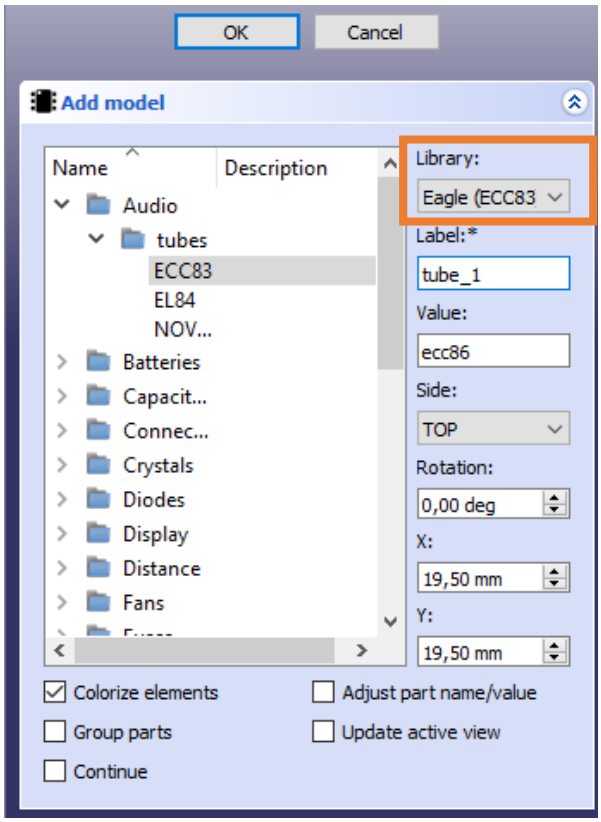
1. Click button **Add model**, new form will appear



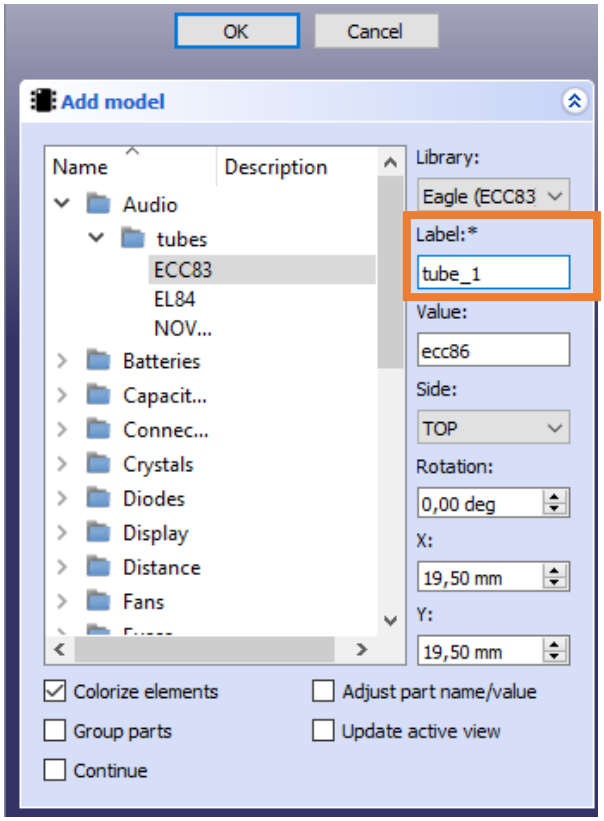
2. Select package – model type



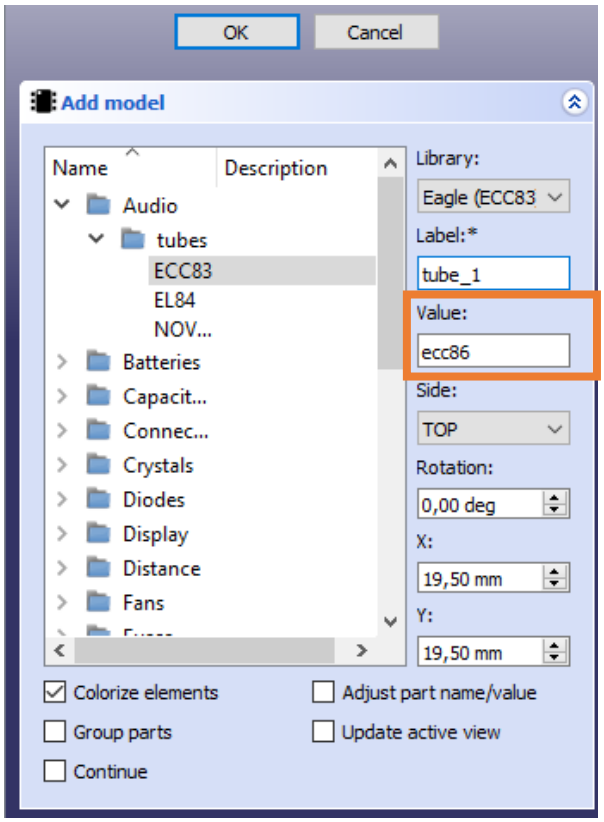

3. Choose from drop-down list, from which library script should take settings



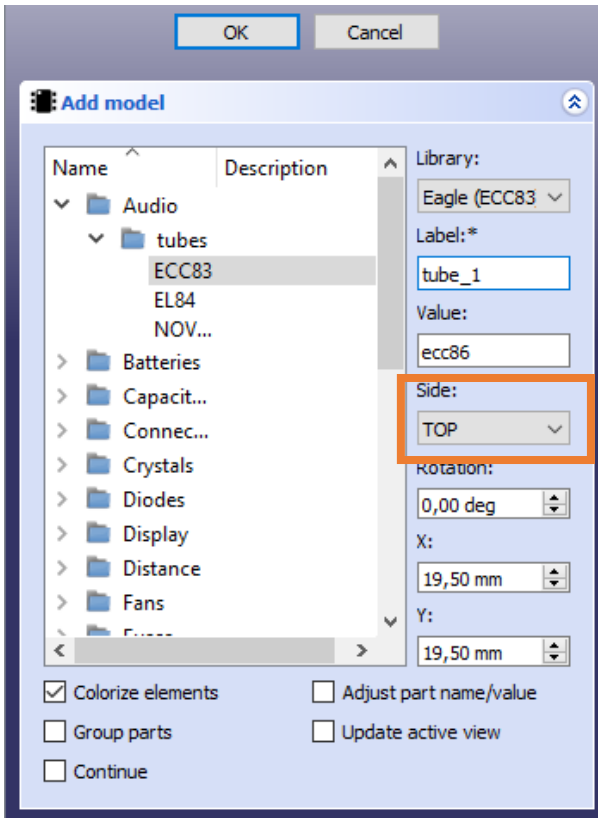
4. Set component name



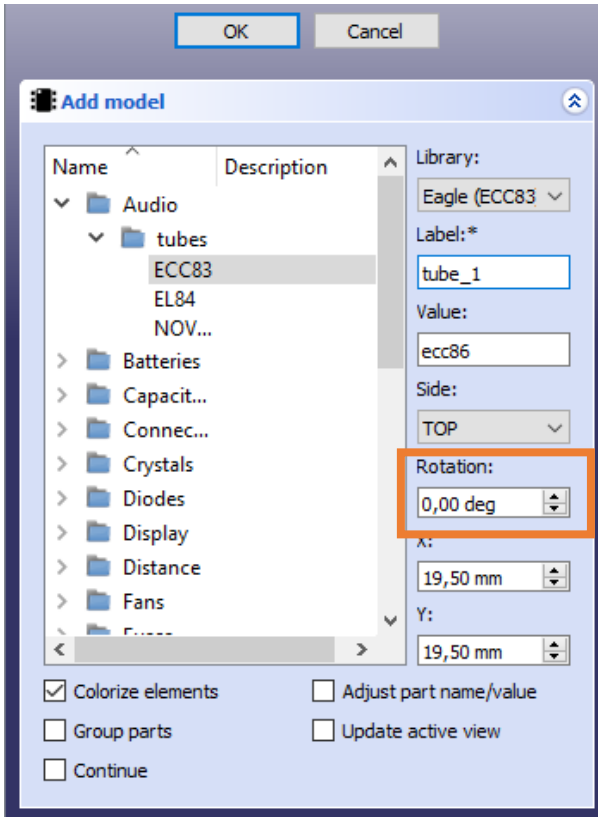

5. Set value



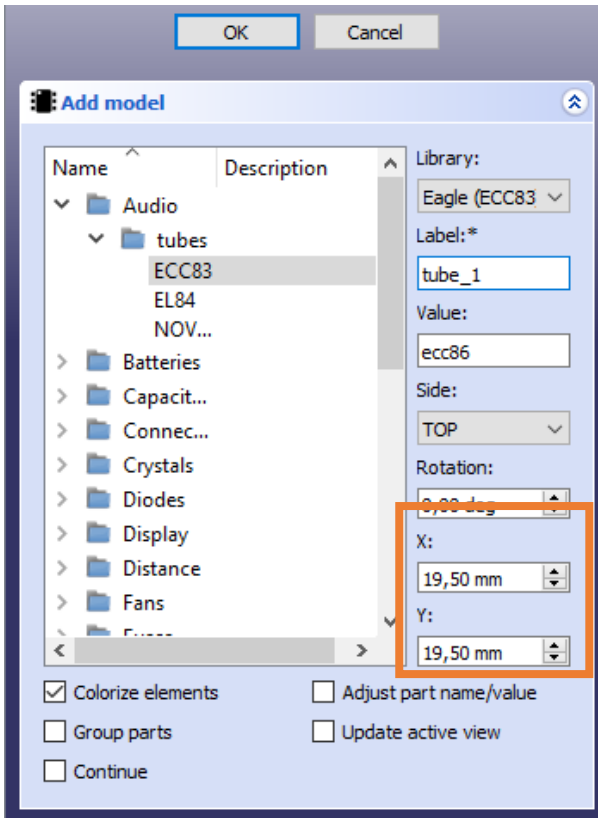
6. Choose side for new component (on board)



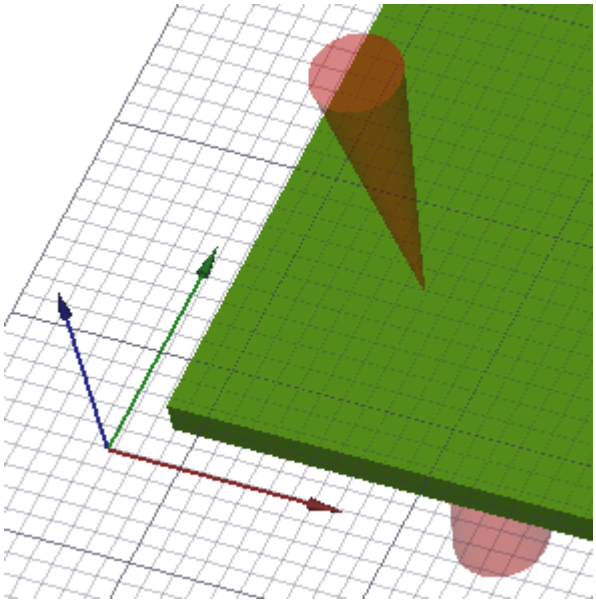

7. Set rotation value (rotation around Z axis)



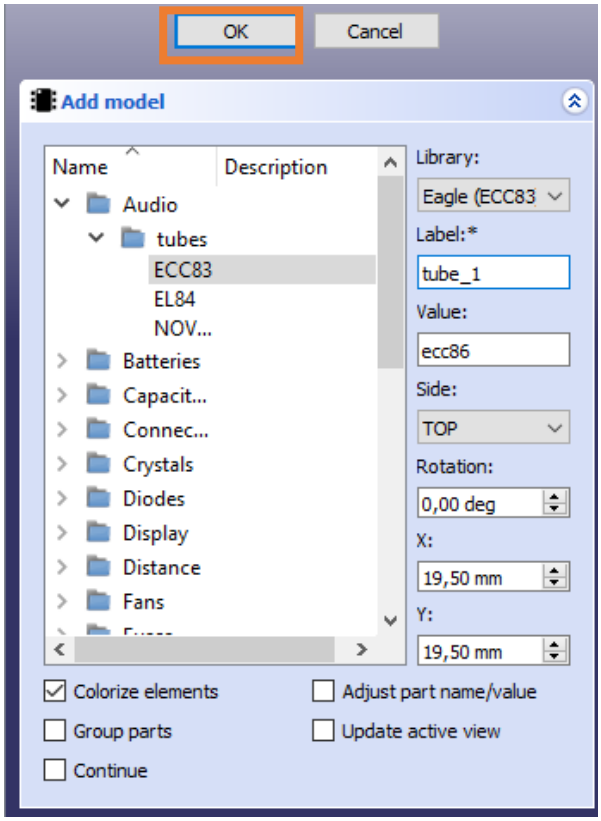
8. Set placement (X, Y coordinates) -–according to global 0

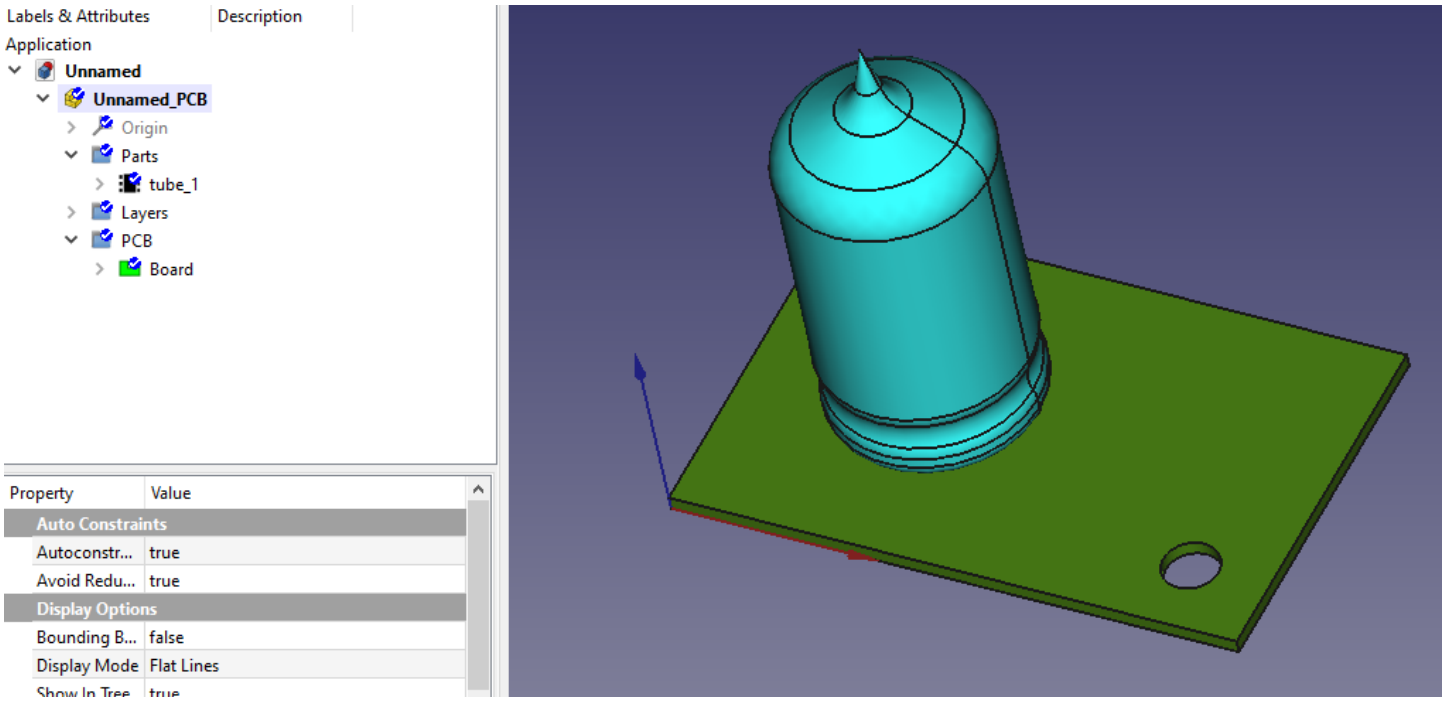



Actual position (model center) is representing in 3D view by red 'arrow'



9. Click Ok button to finisz

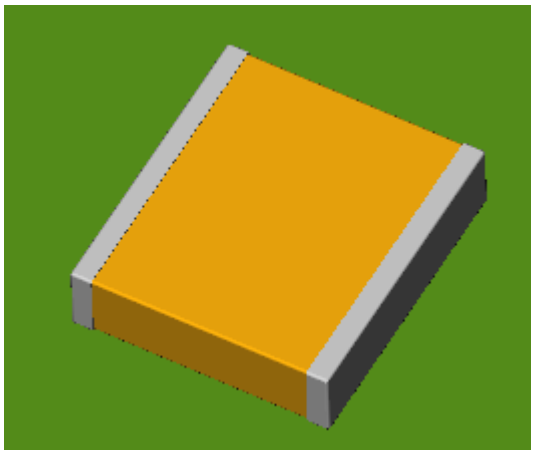


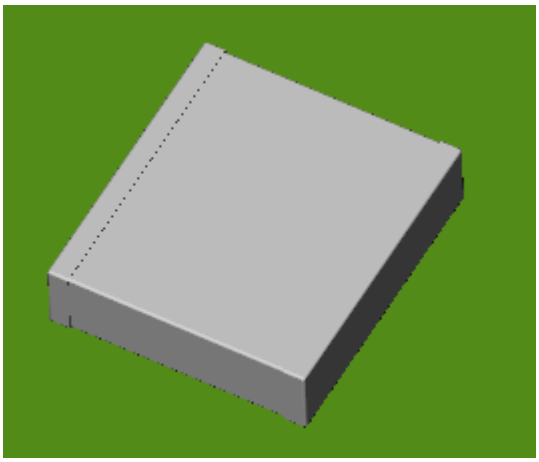
## Adding models – additional settings

Add model tab contain five configuration options:

1. Colorize elements: there is possibility to add models in two modes – with colors and without.



Model added with option 'Colorize elements'

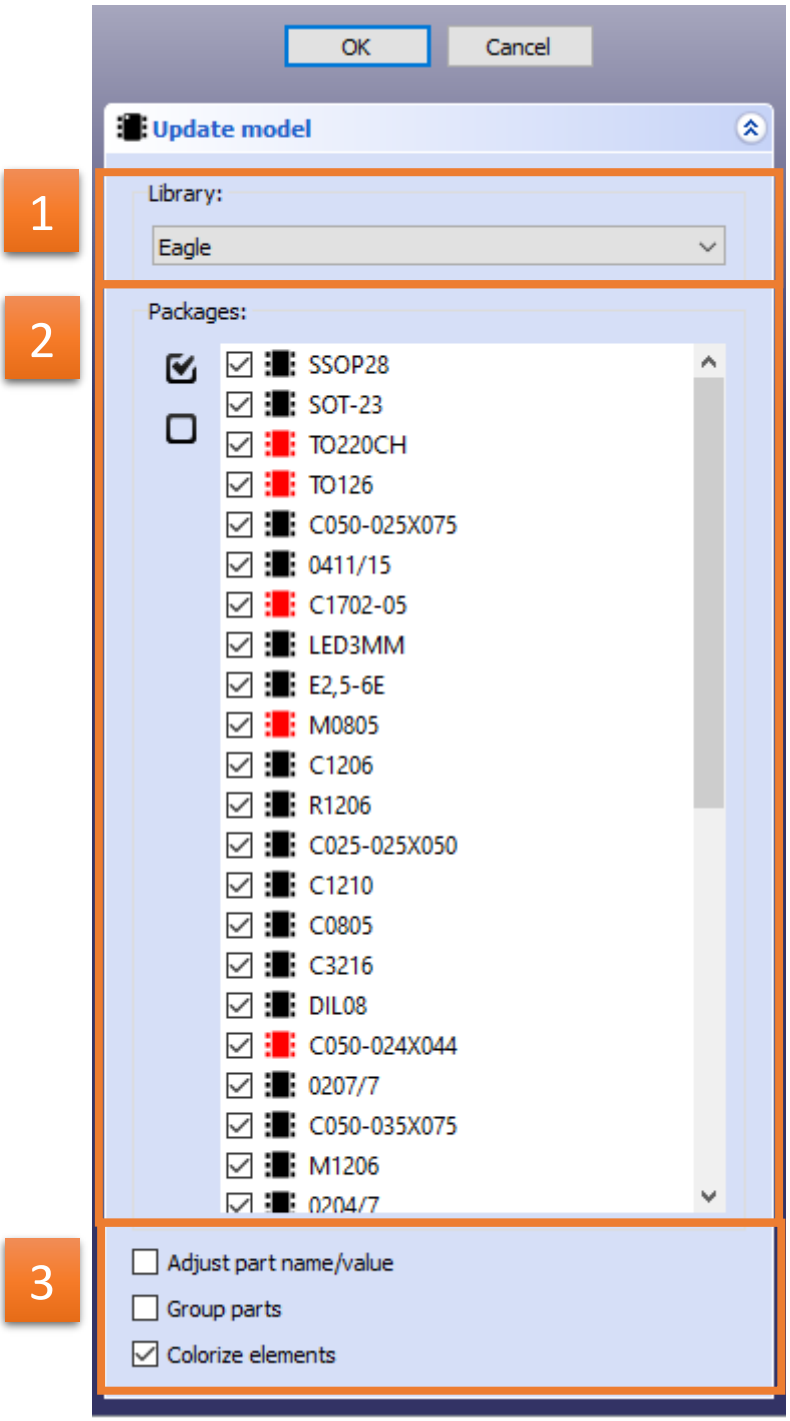


Model added without option 'Colorize elements'

2. Adjust part name/value – set Name/Value annotation values according to settings set in database. For more details check Adjust part name/value section.
3. Update active view: view in 3D window will automatically switches between TOP/BOTTOM view, dependency which side will be chosen.
4. Group parts: grouping parts in tree according to Categories. For more details check Grouping parts section
5. Continue: normally after click Ok button Add modal window disappears, to avoid that (you want to add more than one object) just mark this option.

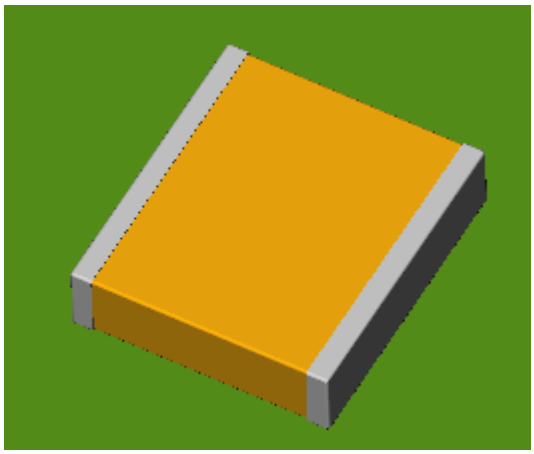
# UPDATING MODELS

Update models window will reload/load 3D model/settings for used in project components.

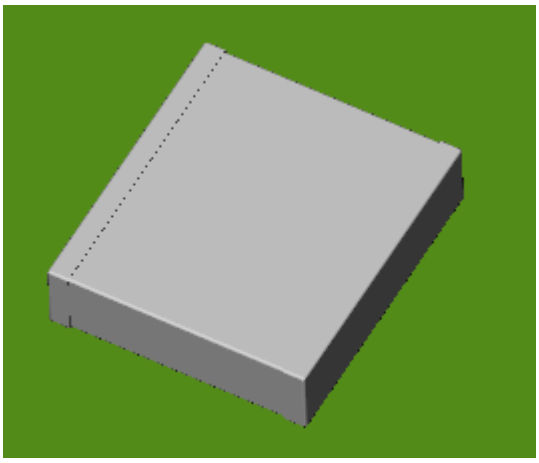


Update models tab contain three sections:

1. Library: during update process, script will search settings (eg. X, Y, Z values) in specific library,
2. Packages: contain listbox with used in project components. Checked checkbox next to model type mean that this part will be updated.
3. Configuration options:
   - Adjust part name/value – set Name/Value annotation values according to settings set in database. For more details check Adjust part name/value section
   - Group parts: grouping parts in tree according to Categories. For more details check Grouping parts section
   - Colorize elements: there is possibility to add models in two modes – with colors and without (grayscale)



Model added with option 'Colorize elements'



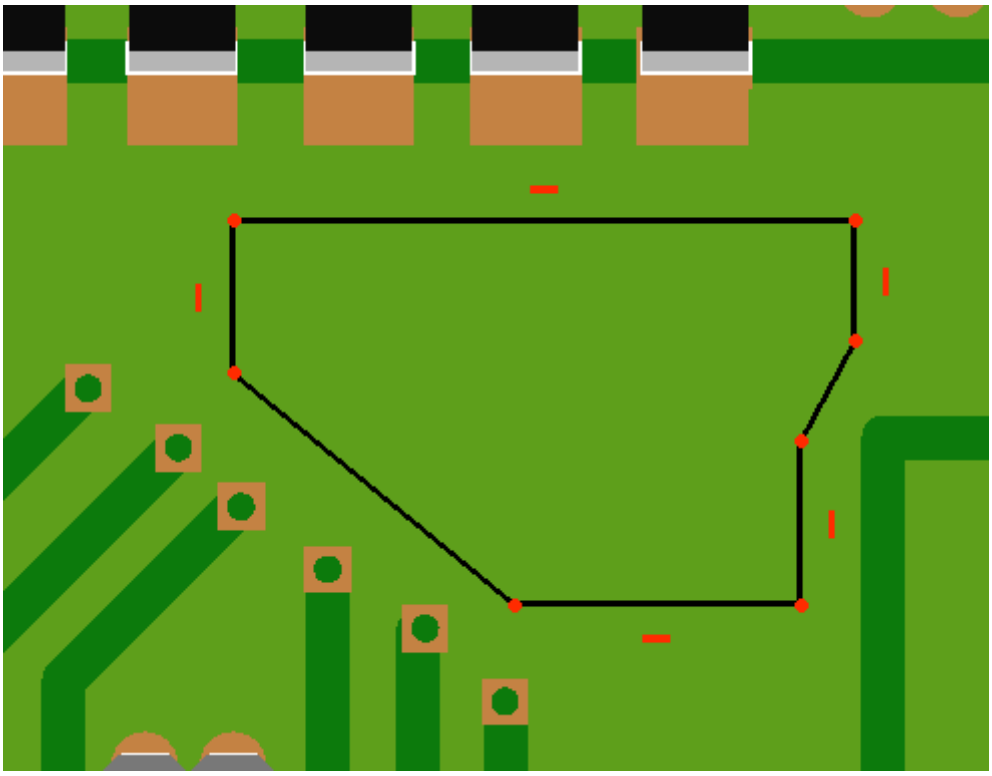
Model added without option 'Colorize elements'

**When selected component does not appear in specified library, model will be not updated.**

# CREATING CONSTRAINST AREAS

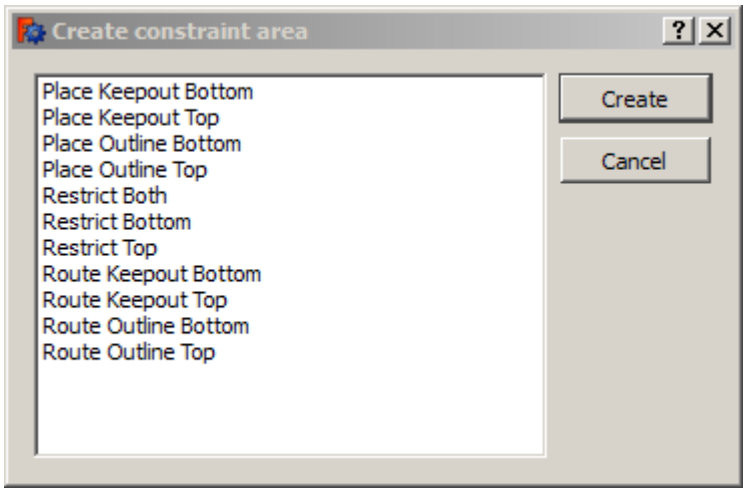
In this section you will find informations how to create a constraint area. A constraint area is a 'object' represent area reservation for different purposes.

1. Create Sketcher with contour of the constraint area



**A shape created in Sketcher must create a closed area.**

2. Select just created sketcher and click Create Constraint area button




3. Choose constraint area type

   Available constraint area types:

   - Place Keepout Bottom:
   - Place Keepout Top:
   - Place Outline Bottom:
   - Place Outline Top:
   - Restrict Both:
   - Restrict Top:
   - Restrict Bottom:
   - Route Keepout Bottom:
   - Route Keepout Top:
   - Route Outline Bottom:
   - Route Outline Top:

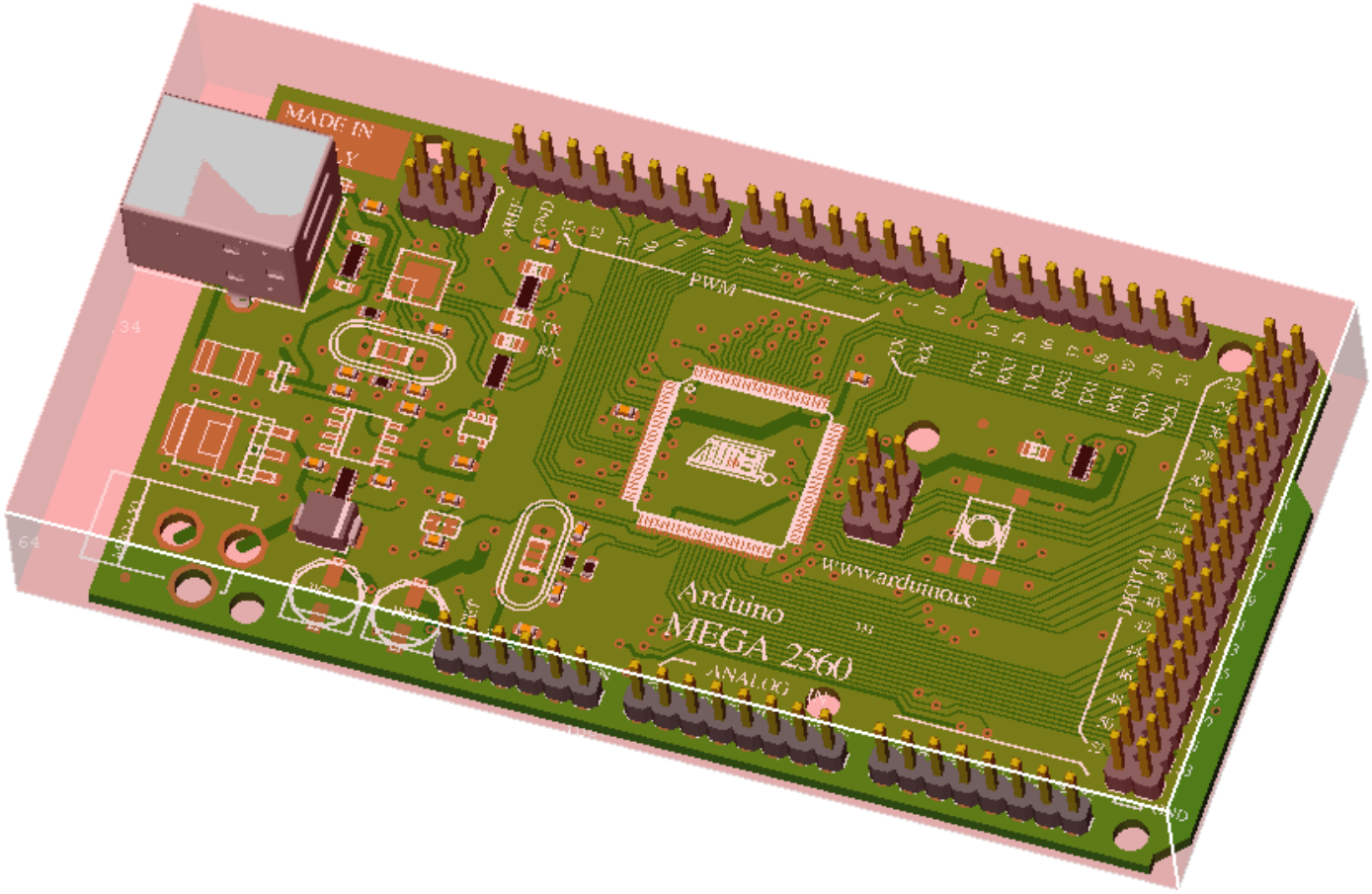
4. Click **OK** button to finish

# GENERATING BOUNDING BOX

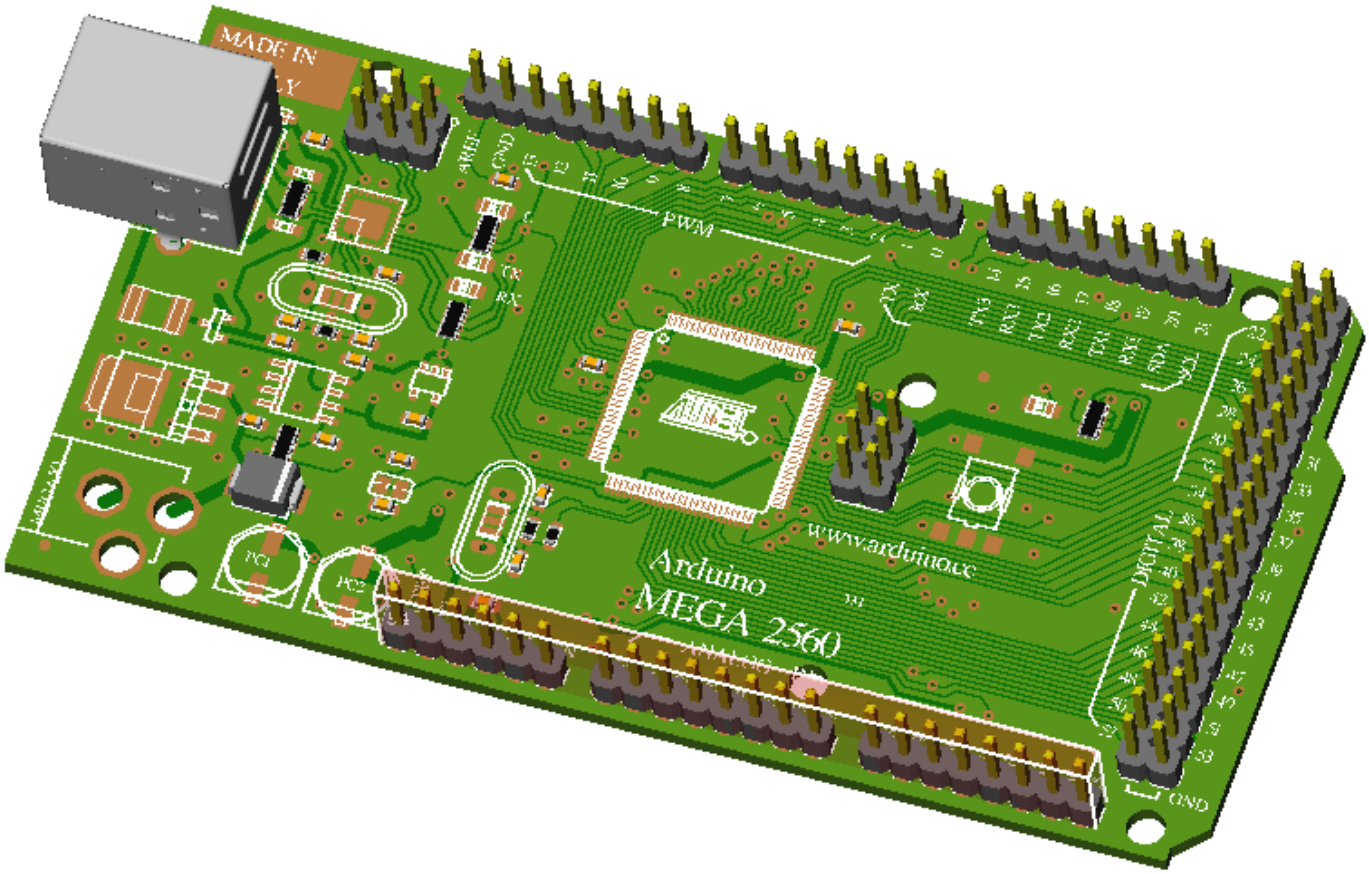
Bounding box is a smallest cuboid completely surrounds the object.

Printed Circuit Board workbench contain two function to generate bounding box:

- Bounding box – generate box for all board (board, parts, paths),
- Bounding box from selection – generate box for selected components.




Bounding box generated for whole board




Bounding box generated for selected components

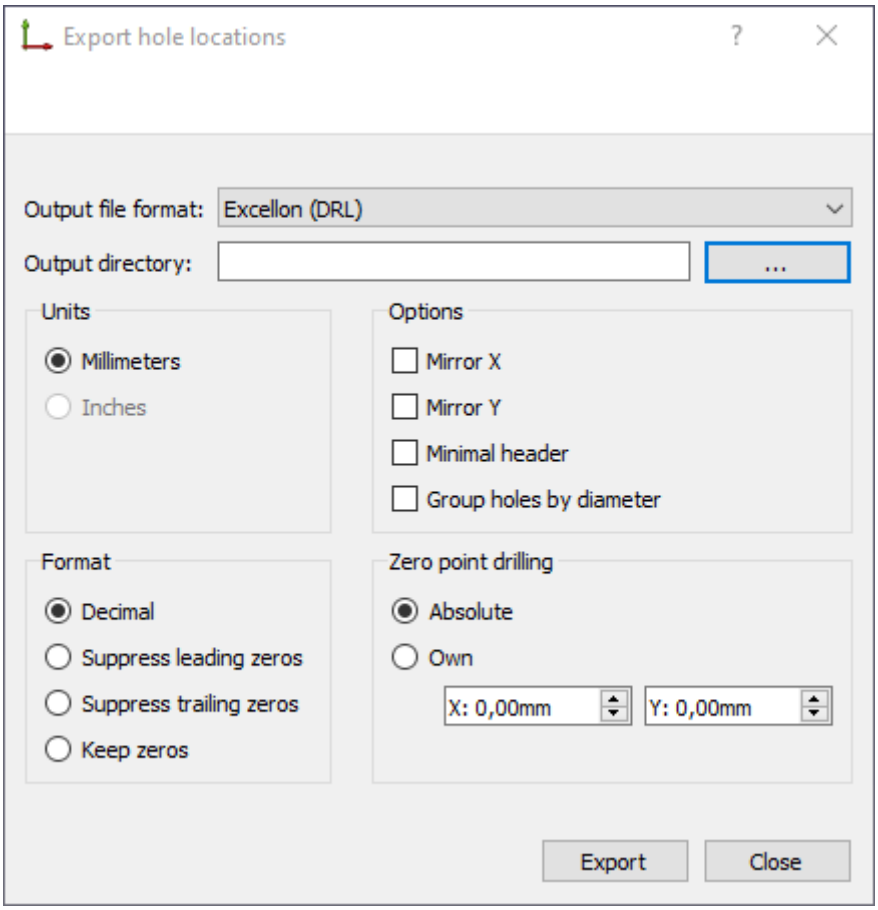
Generated boxes are normal cubes so it is possible to work with them in FreeCAD.

**You can generate as many bounding boxes, as you need..**

# CREATING SECTION CUTS

# EXPORTING HOLE LOCATIONS

Option Export hole locations allow You to export holes list to one of supported file formats.




Export hole location contains a number of settings that allow you to obtain the desired output file format:

1. Output file format:
   - Comma Separated Values (*.csv)
   - Text File (*.txt)
   - HyperText Markup Language (*.html)
   - Excellon (DRL)
2. Output directory: set path where file will be saved

3. Units:
   - Millimeters: measure Everything in Metric , default value
   - Inches: measure Everything in Inches, disabled option
4. Format: choose format, in which values will be saved in file

   Base value: 12.5[mm]
   - Decimal: without changes, value = 12.5
   - Suppress leading zeros: value = 12500
   - Suppress trailing zeros: value = 00125
   - Keep zeros: value = 0012500
5. Zero point drilling
   - Absolute: base point for drilling is set in global 0, 0
   - Own: set new base point for drilling
     - X: X value for new base point for drilling
     - Y: Y value for new base point for drilling

1. Extra options
   - Mirror X: multiply X value by -1
   - Mirror Y: multiply Y value by -1
   - Minimal header: set whether extra data (project name, date, format) will be saved in to output file

```
Drill file
Project: sterownik
Date: 2015-04-25 16:02:37.990862
Unit: mm
Format: Decimal
Zero point drilling: Absolute (0 x 0)
```

   - Group holes by diameter: some output formats support grouping for holes by diameter

| Diameter | X | Y |
|---|---|---|
| 0.5 | 34.3 | 35.6 |
| 0.5 | 22.5 | 29.5 |
| 0.5 | 31.7 | 35.6 |
| 1.0 | 14.9 | 2.6 |
| 1.0 | 85.2 | 11.3 |
| 1.0 | 94.31 | 70.25 |
| 1.0 | 98.09 | 70.25 |
| 1.0 | 94.31 | 64.45 |
| 1.0 | 98.09 | 64.45 |
| 1.0 | 65.61 | 70.25 |
| 1.0 | 69.39 | 70.25 |
| 1.0 | 65.51 | 64.35 |
| 1.0 | 69.29 | 64.35 |
| 3.0 | 10.0 | 18.5 |
| 3.0 | 90.0 | 18.5 |

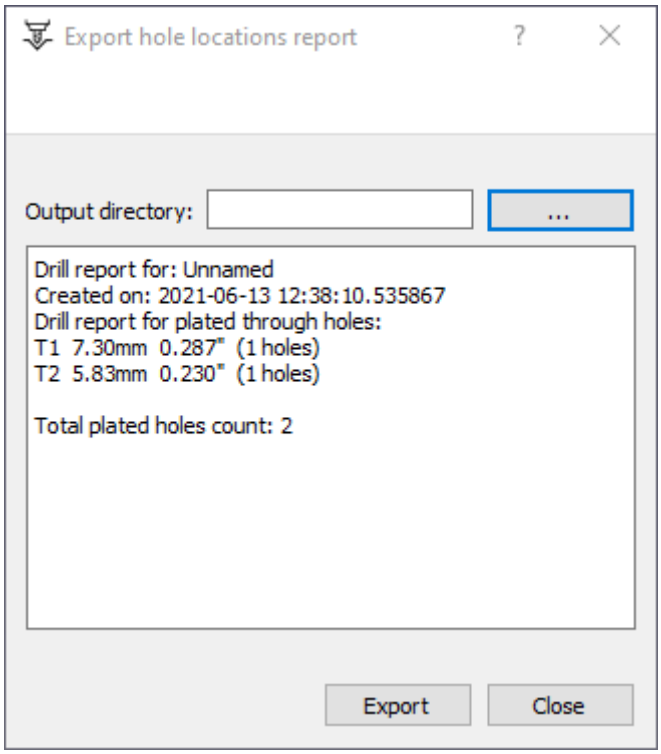
Exported list without option 'Group holes by diameter'.

| Diameter | X | Y |
|---|---|---|
| 0.5 | | |
| | 34.3 | 35.6 |
| | 22.5 | 29.5 |
| | 31.7 | 35.6 |
| 1.0 | | |
| | 14.9 | 2.6 |
| | 85.2 | 11.3 |
| | 94.31 | 70.25 |
| | 98.09 | 70.25 |
| | 94.31 | 64.45 |
| | 98.09 | 64.45 |
| | 65.61 | 70.25 |
| | 69.39 | 70.25 |
| | 65.51 | 64.35 |
| | 69.29 | 64.35 |
| 3.0 | | |
| | 10.0 | 18.5 |
| | 90.0 | 18.5 |
| 0.8 | | |
| | 97.4 | 27.6 |
| | 97.5 | 23.2 |

Exported list with option 'Group holes by diameter'.

# EXPORTING HOLE LOCATIONS REPORT

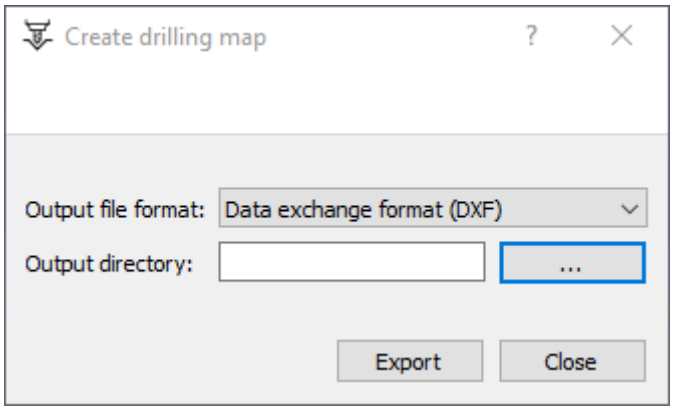
Option Export hole locations report allow You to export report about needed, for drill process, tools.




**Output file have '*.rpt' extension.**

# CREATING DRILLING MAP

Option 'Create drilling map' allow You to create 2D representation of board with marked drilling points. Holes are splitted by diameter – each diameter value is represented by different symbol and color.

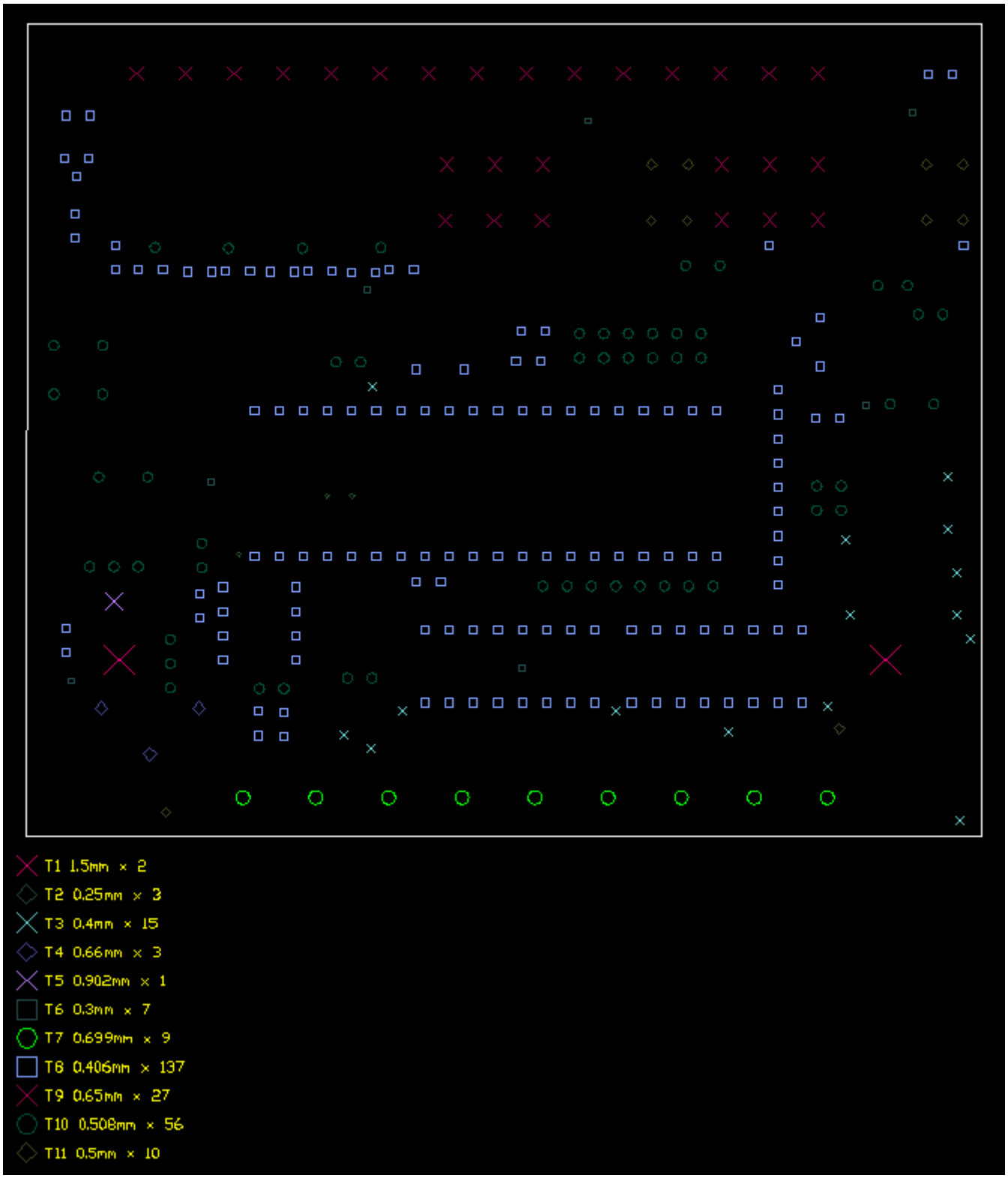



Supported formats:

- DXF: Data exchange format (*.dxf),
- SVG: Scalable Vector Graphics (*.svg).

**Output file name is the same as project in FreeCAD.**

**File extension depends from selected output format.**

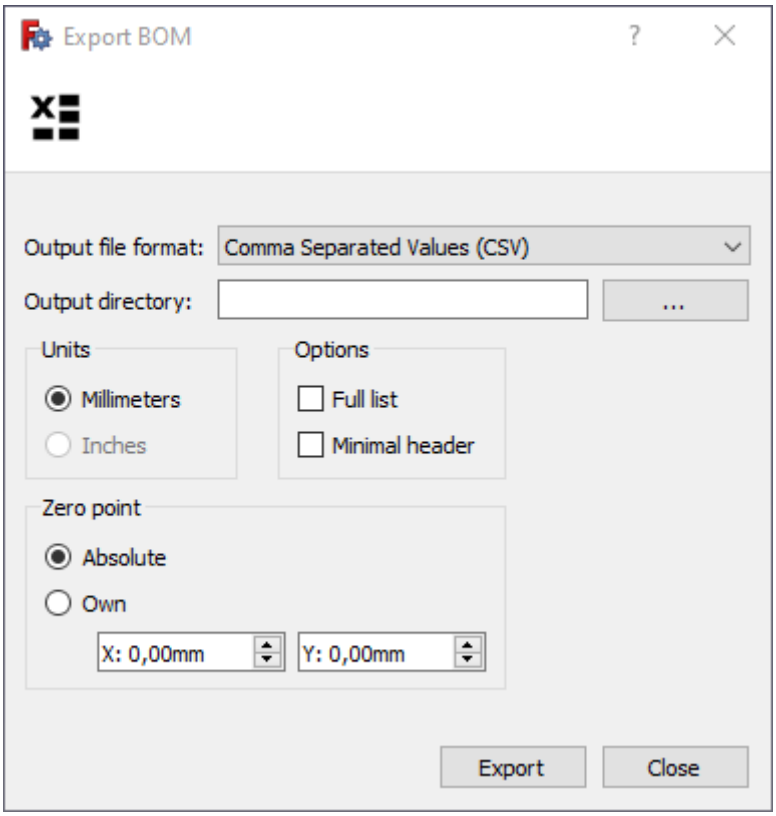



Format in the legend: T1 1.5mm x 2

- T1: tool number,
- 1.5mm: hole dimater in [mm],
- 2: number of holes with same diameter.

# BOM

Option Export BOM allow You to export bill of material list to one of supported file formats.




Export hole location contains a number of settings that allow you to obtain the desired output file format:

2. Output file format:
   - Comma Separated Values (*.csv)
   - Text File (*.txt)
   - HyperText Markup Language (*.html)
3. Output directory: set path where file will be saved
4. Units:
   - Millimeters: measure Everything in Metric , default value
   - Inches: measure Everything in Inches, disabled option

5. Zero point drilling
    - Absolute: base point for drilling is set in global 0, 0
    - Own: set new base point for drilling
        - X: X value for new base point for drilling
        - Y: Y value for new base point for drilling
6. Extra options
    - Minimal header: set whether extra data (project name, date, format) will be saved in to output file

```
Drill file
Project: sterownik
Date: 2015-04-25 16:02:37.990862
Unit: mm
Format: Decimal
Zero point drilling: Absolute (0 x 0)
```

    - Option 'Full list' allow You to generate complex report for used components.

| | Package | Value | Part |
|---|---|---|---|
| 1 | 1X02 | | E$33, E$17, E$19 |
| 2 | 2X06 | | E$24 |
| 3 | TL1105SP | | S1 |
| 4 | 1206 | | LED1, LED3, LED2, LED5, LED4, LED7, LED6, LED9, LED8, LED19, LED18, LED11, LED10 |
| 5 | H2M09ST | | X1 |
| 6 | 1X08 | | E$32 |
| 7 | TUXGR_16X2_R2 | | DIS1 |
| 8 | 2X02 | | E$20 |
| 9 | R2512 | 150R | R44, R58, R60, R40, R52, R50, R24, R55 |
| 10 | R2512 | 1k2 | R17, R7, R3, R22, R46, R34, R37, R31 |
| 11 | R0805 | 1k | E$25 |
| 12 | R1206 | 150R | R59, R12, R9 |
| 13 | | | |

Exported list without option 'Full list'.

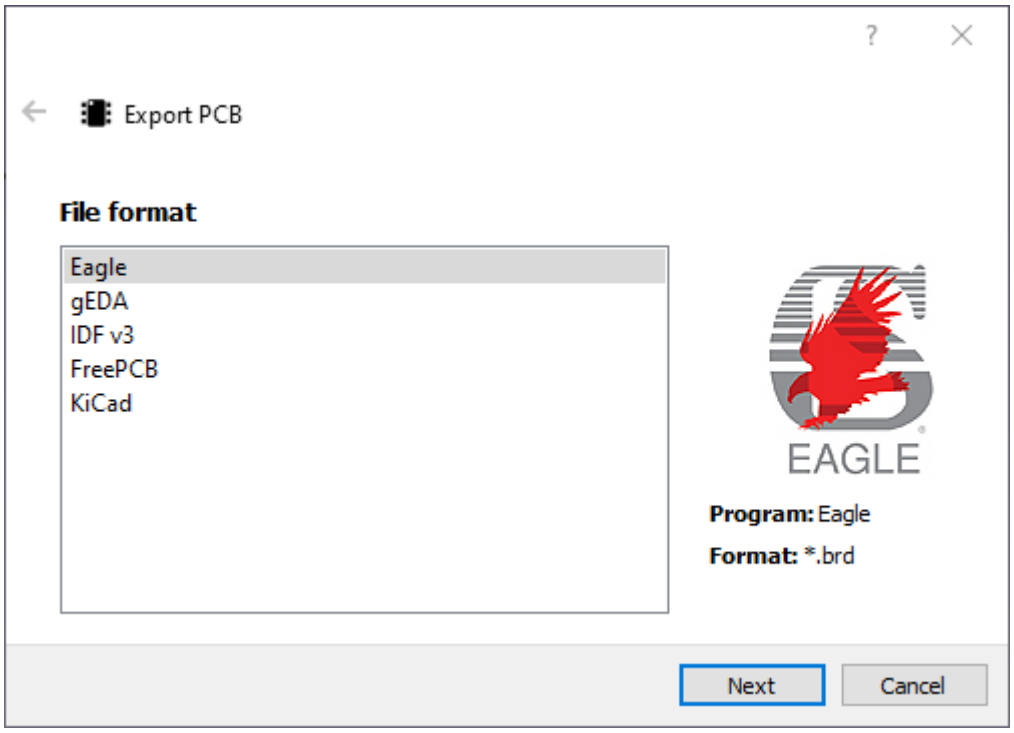
| | ID | Package | Value | X | Y | Rotation | Side | Quantity |
|---|---|---|---|---|---|---|---|---|
| 1 | | | | | | | | |
| 2 | X2 | PN61729 | | 3.81 mm | 38.1 mm | 270 deg | TOP | 1 |
| 3 | POWER | 1X06 | | 39.37 mm | 2.54 mm | 0 deg | TOP | 1 |
| 4 | Z1 | CT/CN0603 | PGB1010604 | 12.065 mm | 35.56 mm | 0 deg | TOP | 2 |
| 5 | Z2 | CT/CN0603 | PGB1010604 | 12.065 mm | 40.64 mm | 0 deg | TOP | 2 |
| 6 | JP1 | 1X01 | | 93.98 mm | 50.8 mm | 0 deg | TOP | 4 |
| 7 | JP3 | 1X01 | | 93.98 mm | 7.62 mm | 0 deg | TOP | 4 |

Exported list with option 'Full list'.

# CENTROID

# EXPORTING BOARD

Export option allow you to save created/modified board in FreeCAD to one of supported file formats.

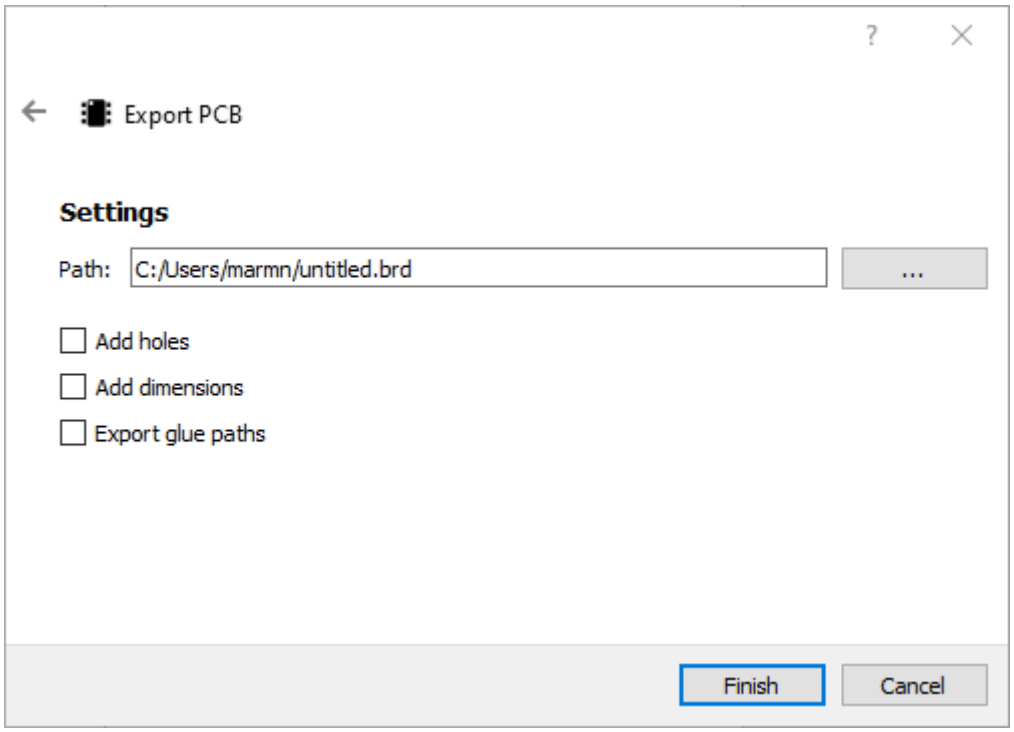



First tab in export window allows You to choose export file format.

Supported files:

- Eagle (*.brd),
- FidoCadJ (*.fcd),
- KiCad (*.kicad_pcb),
- gEDA (*.pcb),
- Razen (*.rzp).

After choosing file format click Next to move to settings section where You can set which parts of PCB will be exported and where to save new file.




Available options depend on the selected file format.

**Default settings you can set in Preferences tab 'Customizing Workbench'.**

Clicking Finish button will end Export process and script will create new file according to chosen settings.

## Unit system

During board export process units are changed to millimeters [mm].

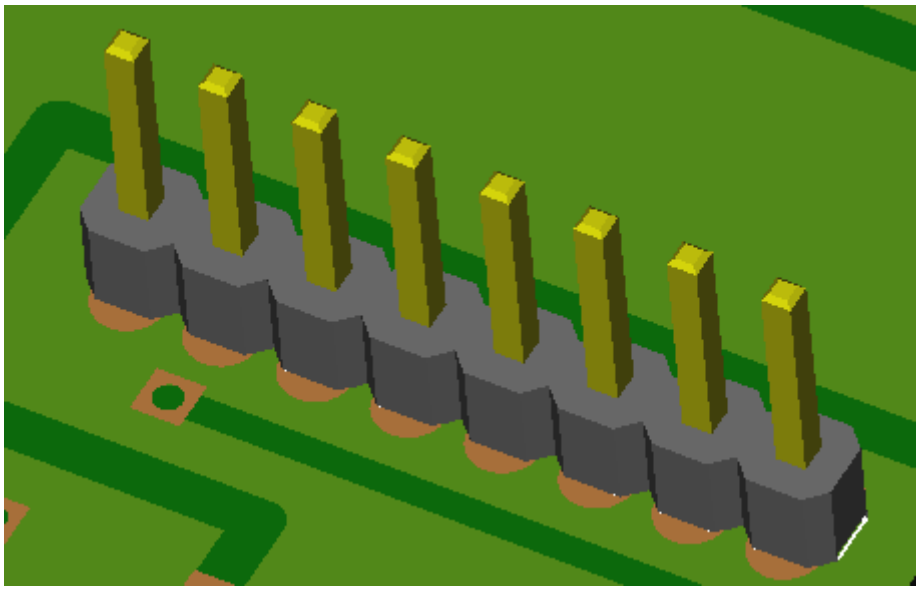
# VIEW OPTIONS

# DISPLAY MODES

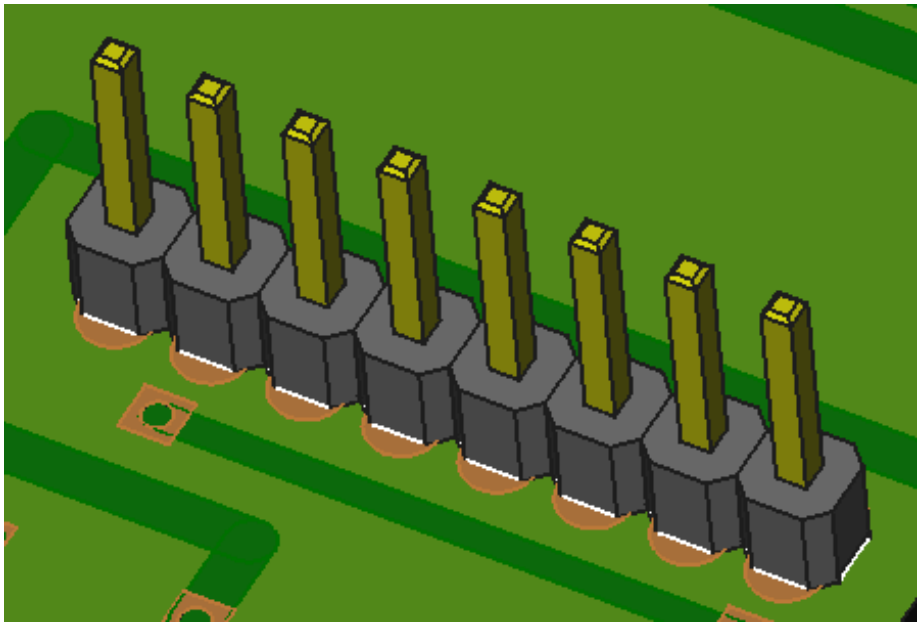
The Display Modes function allows you to quickly and easily change the display representation of shapes in your project.

Available types:

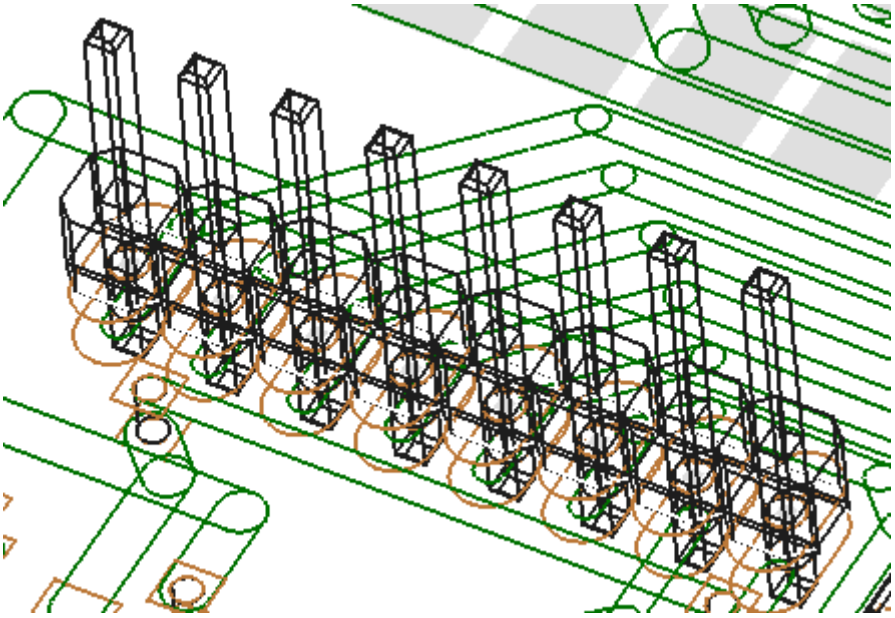
1. Shaded: border lines are hidden.



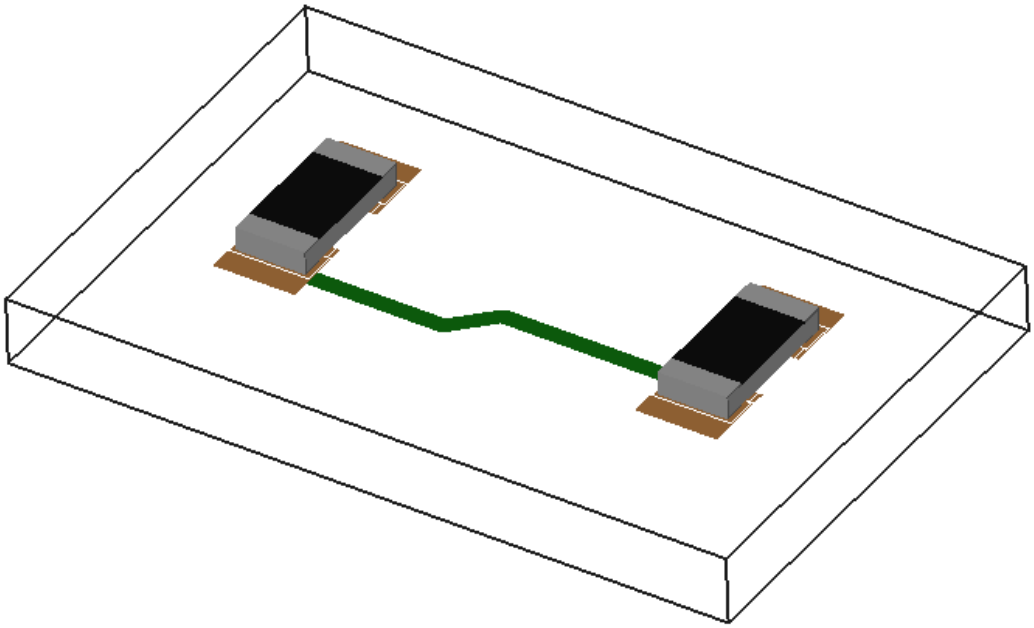
2. Flat lines: surfaces and border lines are displayed in one time.



3. Wireframe: only border lines are displayed.

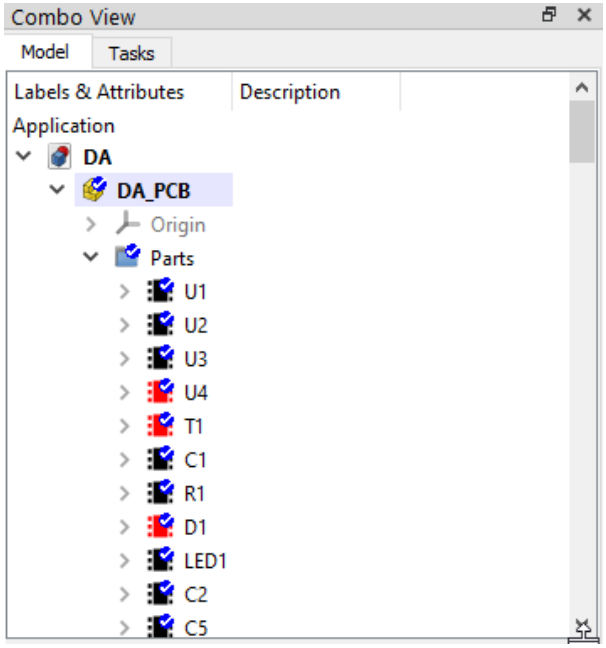


4. Internal View: for board only border lines are displayed, rest is displayed in Flat lines mode

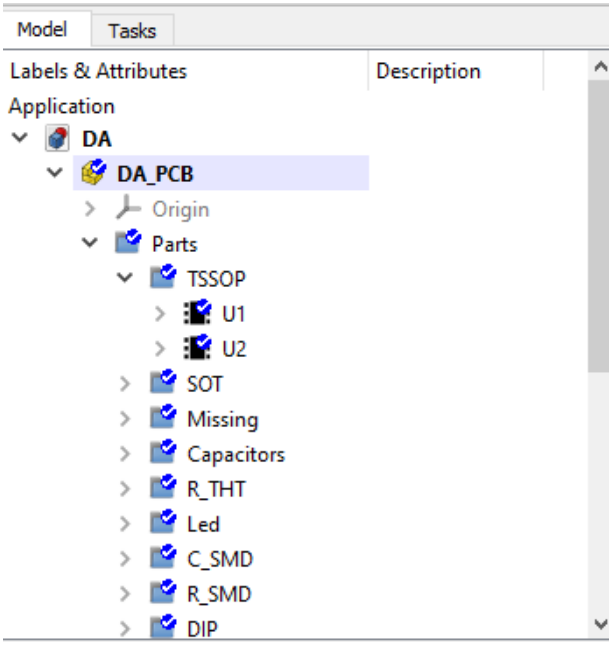


# GROUPING PARTS

These options allow you to group/ungroup parts according to the categories they belong to (parameters stored in the database).




Ungrouped parts



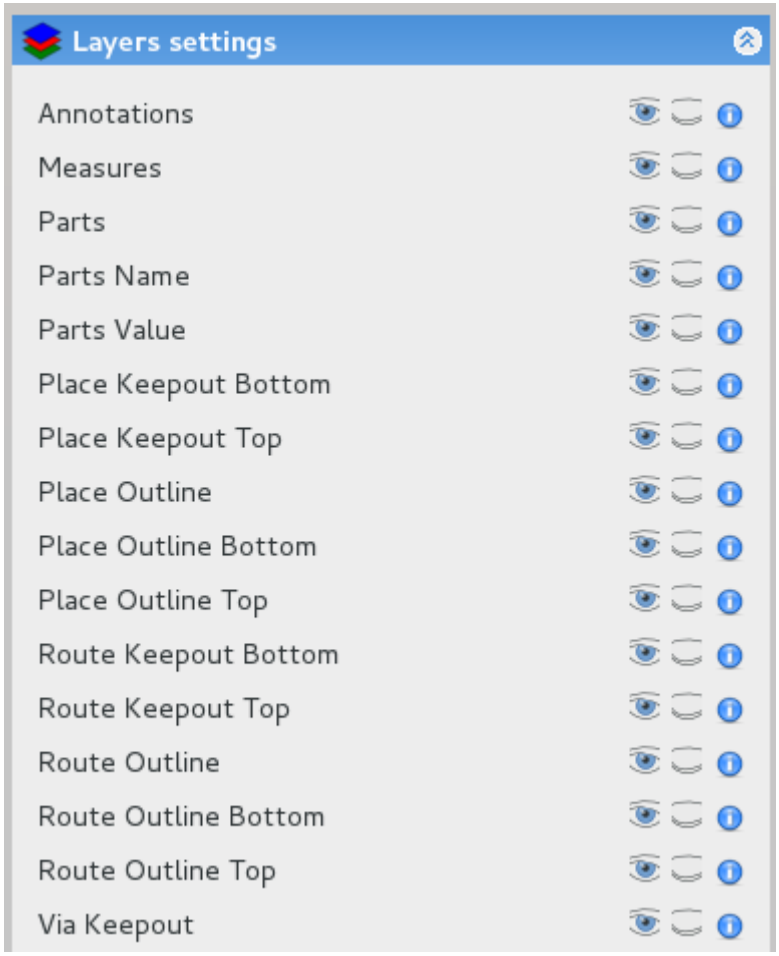

Grouped parts

These options are also available in:

- open/import window,
- update parts window,
- add new model window.

# LAYERS

The layer settings window helps you manage the currently displayed layers of the board. The layer settings window appears on the Task tab.



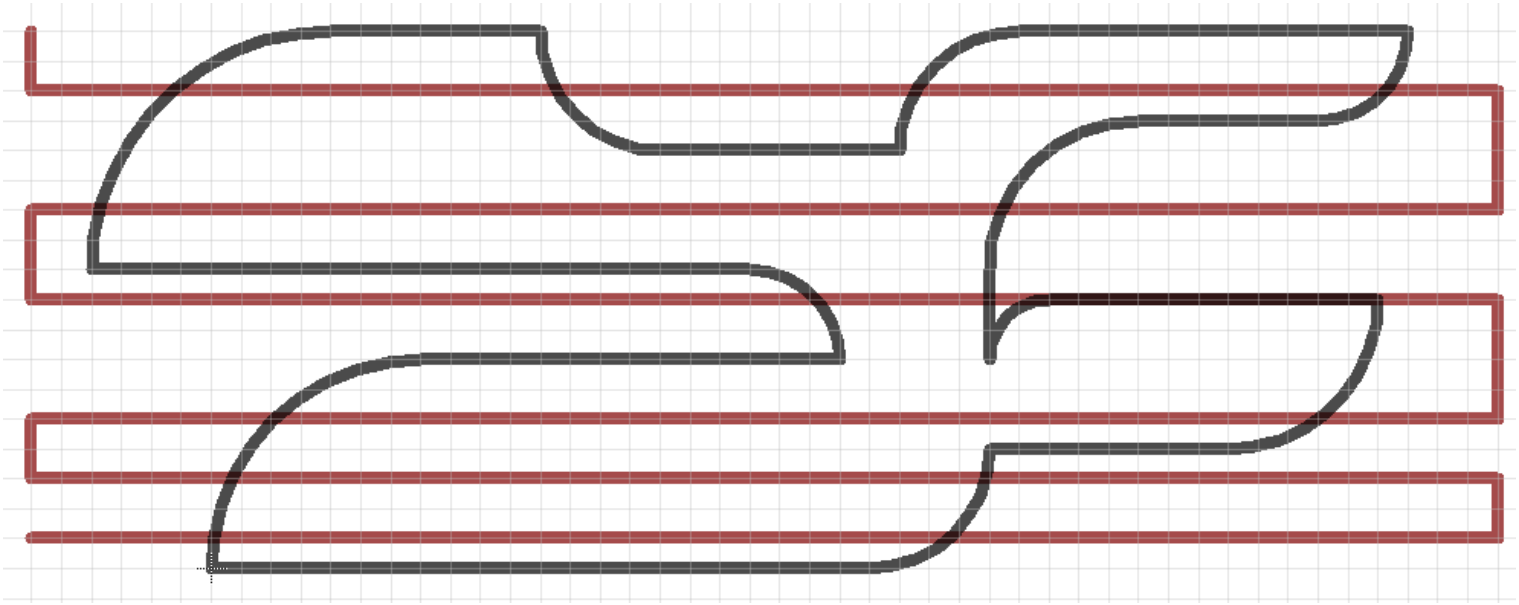

Each line consists of four parts:

- Layer name,
- Button Show All – show all objects of this type,
- Button Hide All – hide all objects of this type,
- Information button – display information about layer.

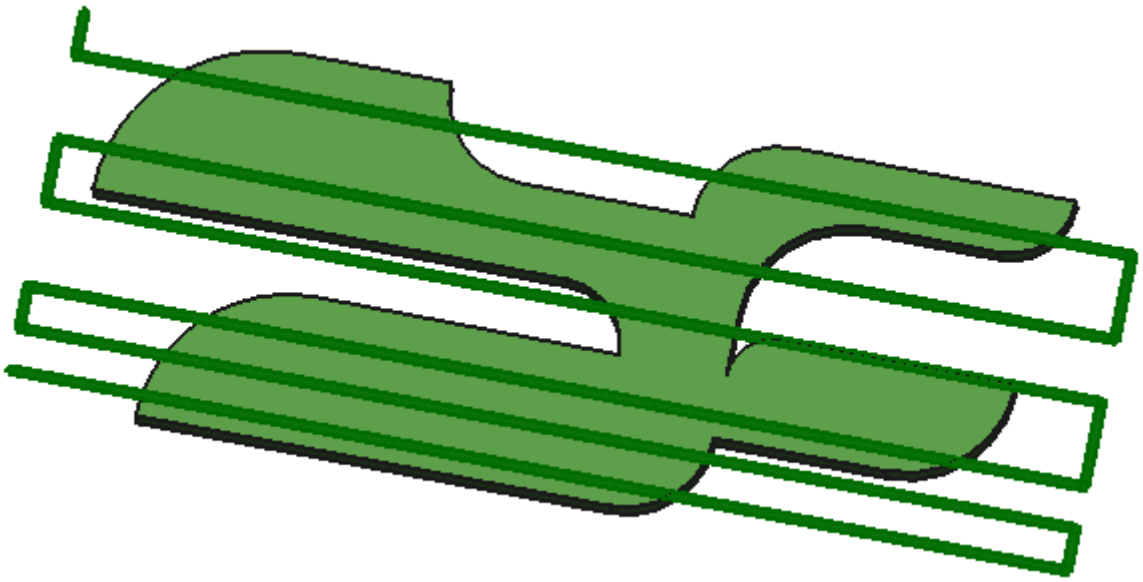
# CUT TO BOARD OUTLINE

Sometimes it is necessary to display board like it will look after manufacturing. To do this just use option 'Cut to Board Outline'. Function will automatically blank/display all layers/paths that are outside of the board.

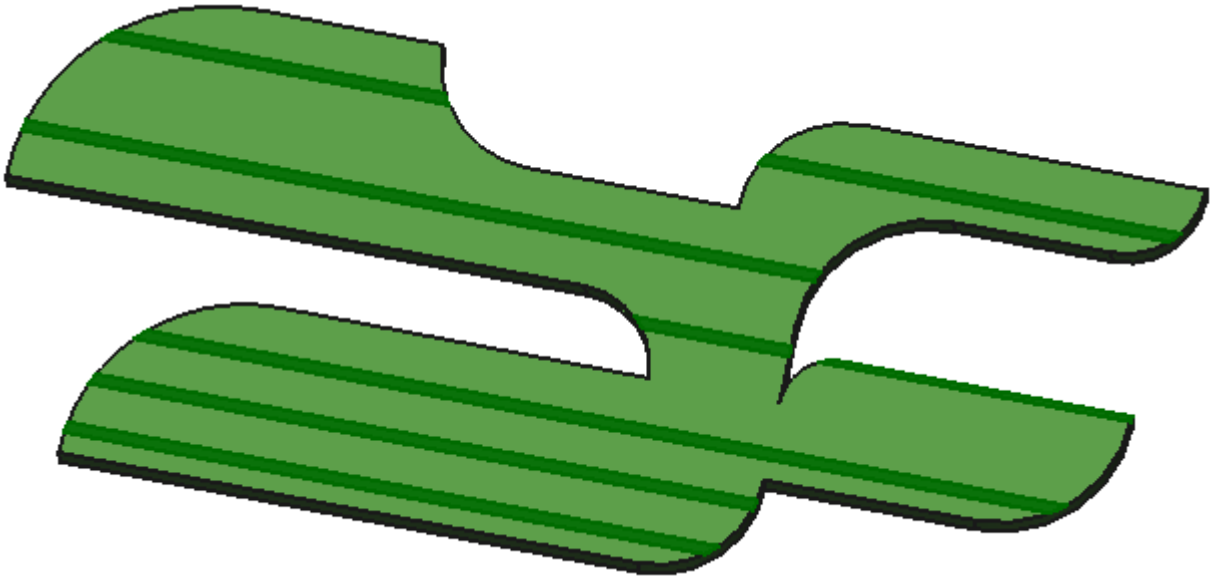
**Example**



Board created in Eagle



Cut to Board Outline = OFF



Cut to Board Outline = ON

# HOLES SETTINGS

# SIGNALS MARKING

# RENDERS

# KERKYTHEA

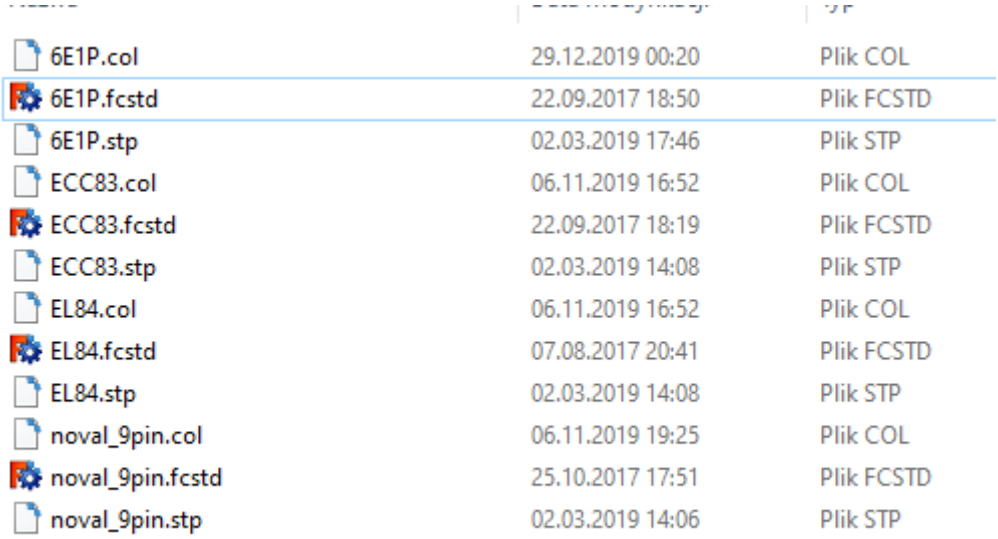
# POV-RAY

# OTHER

# GENERATE MODELS

# *.COL FILES

Mainly script works with models saved in *.stp or *.igs formats, however they are readed/imported directly only once. To seep up board importing for each stp/igs file specific *.col file is generated.



| | | |
|---|---|---|
| 6E1P.col | 29.12.2019 00:20 | Plik COL |
| 6E1P.fcstd | 22.09.2017 18:50 | Plik FCSTD |
| 6E1P.stp | 02.03.2019 17:46 | Plik STP |
| ECC83.col | 06.11.2019 16:52 | Plik COL |
| ECC83.fcstd | 22.09.2017 18:19 | Plik FCSTD |
| ECC83.stp | 02.03.2019 14:08 | Plik STP |
| EL84.col | 06.11.2019 16:52 | Plik COL |
| EL84.fcstd | 07.08.2017 20:41 | Plik FCSTD |
| EL84.stp | 02.03.2019 14:08 | Plik STP |
| noval_9pin.col | 06.11.2019 19:25 | Plik COL |
| noval_9pin.fcstd | 25.10.2017 17:51 | Plik FCSTD |
| noval_9pin.stp | 02.03.2019 14:06 | Plik STP |

File contains four basic informations:

- Line 0: file format | date of last stp/igs file modification
- Line 1: model colors (each surface)
- Line 3 >: models saved if brep format

```
1  3|1551545175.2290237
2  [(0.1882352977991104, 1.0, 1.0, 0.0)]
3  CASCADE Topology V1, (c) Matra-Datavision
4  Locations 0
5  Curve2ds 76
6  1 6.2831853071795862 -5.5999999999999979 -1 0
7  1 -4.1389114358025836e-13 0 -0 -1
8  1 6.2831853071795862 0 -0 -1
9  1 6.2831853071795862 0 -1 0
```

When date saved in first row is lower than last modification of stp/igs file (modification/new model), *.col file will be automatically updated. Similar situation is when *.col file does not exist – new one will be created.

**Generating new *.col file is time consuming**

Sometimes script will load incorrectly 3D model representation/colors, or loaded model is old (relative to stp/igs file). In this situation it is necessary to delete *.col file and generate new one.

# SCRIPTS

There are available few scripts which are helping exporting the boards to FreeCAD.

## Eagle

Directly exporting boards from Eagle to FreeCAD [path: scripts/eagle]

scripts/eagle/ulp/freecad.ulp – copy file to $EAGLEDIR/ulp/

scripts/eagle/scr/freecad.scr – copy file to $EAGLEDIR/scr/

scripts/eagle/bin/freecad.png – copy file to $EAGLEDIR/bin/

In Eagle choose File → Execute Script → freecad.


On Linux to set path to FreeCAD change value of var 'programPath_LIN' in file freecad.ulp.

On Windows to set path to FreeCAD change value of var 'programPath_WIN' in file freecad.ulp.

The script is useful for Eagle versions lower than 7.

# EXAMPLES