Printed Circuit Board Workbench for FreeCAD (PCB)

Flexible Printed Circuit Board Workbench for FreeCAD (FPCB)



marmni (marmni@onet.eu)

Copyright 2013 - 2016

http://sourceforge.net/projects/eaglepcb2freecad/

Index

Introduction.	4
Supported files	
Requirements	
Installation.	
GNU/Linux	
Windows	
Configuration.	
STP file format colors definition	
Accessing the workbench	
Set PCB module as main workbench.	
Menu bars	
Toolbars	
PCB View toolbar	
PCB Settings toolbar	
Displaying toolbars	
Specification tree	
Customizing workbench	
Download models	
Assign models	
Basics setting.	
Adding equivalent package for 3D model	
Adjust part name/value	
Multi model definition for one part	
Open/Import board	
Unit system	
Grouping parts	
Display modes	
Layers	
Update models	
Cut to board outline	
Explode	
Bounding box	
Create new project	
± ¥	
Create PCB Add models	
Add annotations.	
Create glue paths	
Create constraints areas	
Export board	
Export bill of materials (BOM)	
Export hole locations	
Export hole locations report	
Create drilling map	
Create drill center	
Add assembly	
Update assembly	
Collision detection	
Export to Kerkythea	
Export object to POV-Ray	
Objects properties	
File format	
Scripts	
Errors code	
Licence	
Changelog	
ToDo list	84

Drintad	Circuit	Doord	Morkhanah	for	EKOOCAD
Printea	CIRCUIT	Board	Workbench	TOT	FreeCAD

 1	1

Errors......85

INTRODUCTION

[ENG]

Mod allow you to import/create PCB boards in FreeCAD. Scope of mod:

- support for many different layers,
- possible to choose colours, transparency and names for each layer,
- mod allows you to import IGES/STP models with colours,
- possible to show holes/vias independent.

[PL]

Moduł pozwala na importowanie/tworzenie płytek PCB w programie FreeCAD. Możliwości modułu:

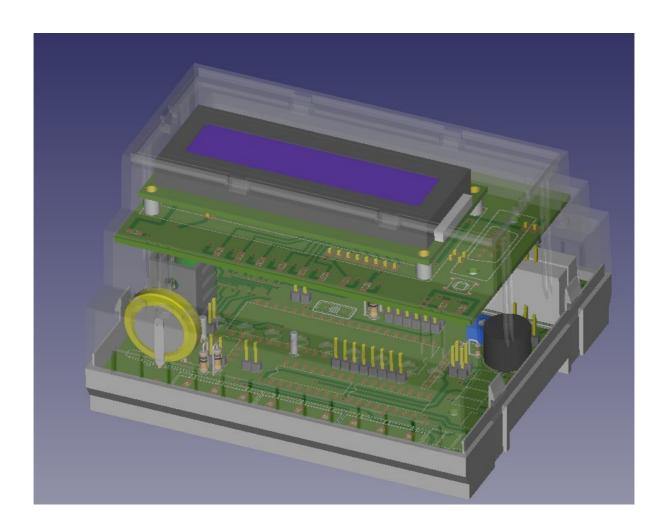
- wsparcie dla wielu różnych warstw,
- wyświetlanie otworów, przelotek niezależnie od siebie,
- możliwość wyboru koloru, przeźroczystości oraz nazwy dla poszczególnych warstw,
- importowanie modeli zapisanych w formacie IGS/STP wraz z kolorami.

Supported files

- Eagle (*.brd),
- Razen (*.rzp),
- FreePCB (*.fpc),
- gEDA (*.pcb),
- FidoCadJ (*.fcd),
- KiCad (*.kicad_pcb),
- IDF v2/v3,
- HyperLynx (*.HYP).

Requirements

FreeCAD-PCB require FreeCAD in version 14.0 or never. Module was tested on Windows and GNU/Linux.



INSTALLATION

Unpack downloaded zip file and copy extracted folder to direction where FreeCAD is installed (subfolder Mod).

GNU/Linux

Example:

FreeCAD path:

~/Programs/FreeCAD

So copy mod to folder

~/Programs/FreeCAD/Mod

You can also copy files to folder ~/.FreeCAD/Mod .

Next change read/write permission to 777. Please don't forget about parameter -R!

Example:

chmod 777 -R PCB

Windows

Example:

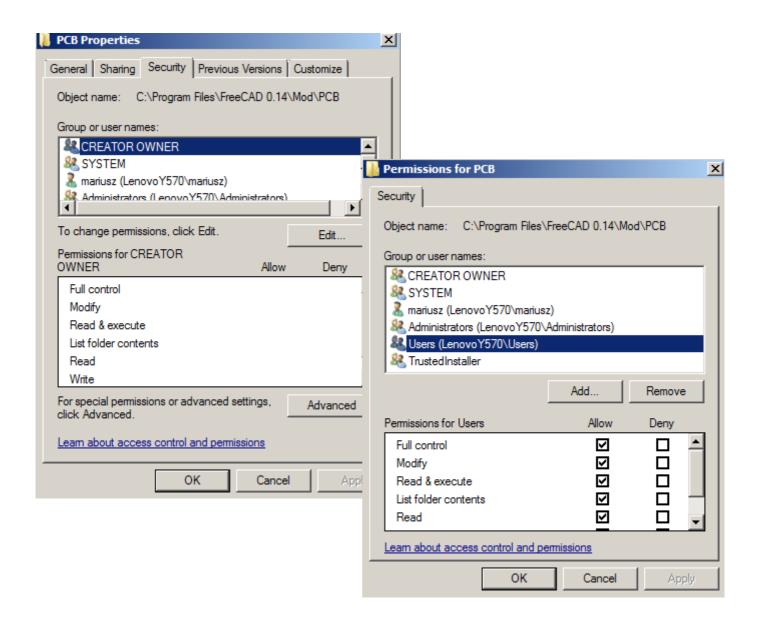
FreeCAD path:

C:/Program Files/FreeCAD-0.14

So copy mod to folder

C:/Program Files/FreeCAD-0.14/Mod

Next change read/write permission for all users. Click right button on folder PCB and choose Properties → Security → Edit → Users and mark all checkboxes under 'Allow' option.



CONFIGURATION

At this moment some settings need to be configured in file PCBconf.py. You can open this file in any text editor (please avoid Notepad).

STP file format colors definition

During loading board, You can meet with error connected with missing STP color definition. To fix that problem just add new color definition in PCBconf.py file in spisKolorowSTP variable.

Example:

```
red
Actual situation:
    spisKolorowSTP = {
        "white": (1.0, 1.0, 1.0),
        "black": (0.0, 0.0, 0.0)
    }
Write to file:
    spisKolorowSTP = {
        "red": (1.0, 0.0, 0.0),
        "white": (1.0, 1.0, 1.0),
        "black": (0.0, 0.0, 0.0)
}
```

Where:

```
"red": (1.0, 0.0, 0.0), => "colorName": (R / 255, G / 255, B / 255)
```

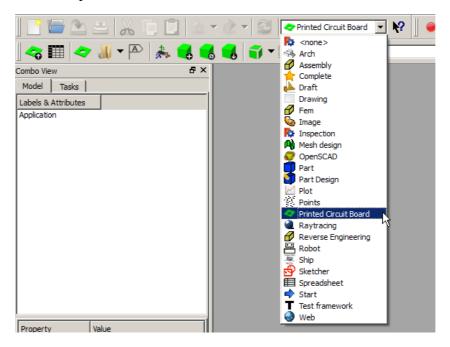


Please do not change anything else in the PCBconf.py file! More configuration options You can find in Customizing Workbench section.

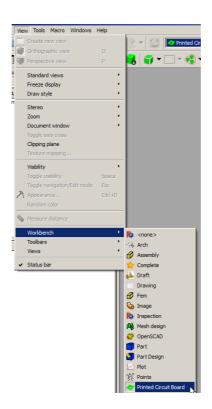
ACCESSING THE WORKBENCH

There are two methods to access to the PCB workbench:

1. In toolbar 'File' locate drop down list and choose 'Printed Circuit Board'.

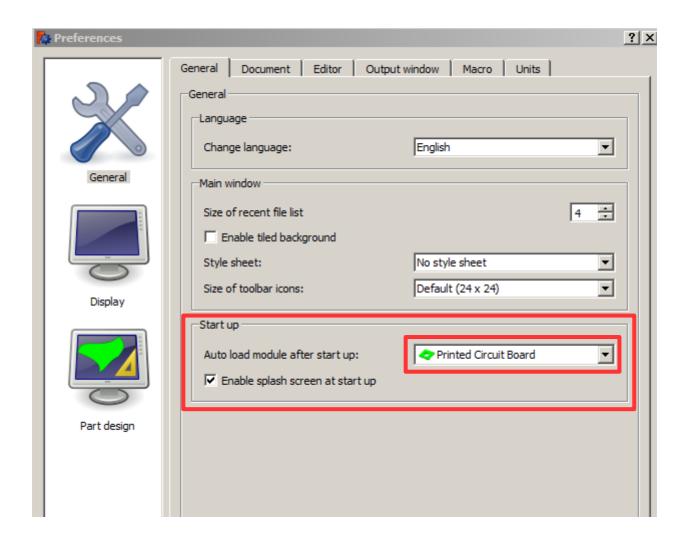


2. From top menu bar choose View \rightarrow Workbench \rightarrow Printed Circuit Board.



Set PCB module as main workbench

There is possibility to set PCB module as main workbench. To do this choose from top menu bar Edit \rightarrow Preferences, in settings window choose General and tab General. In displayed tab You should find 'Start up' section, where You can set which workbench should be loaded after FreeCAD start.



MENU BARS

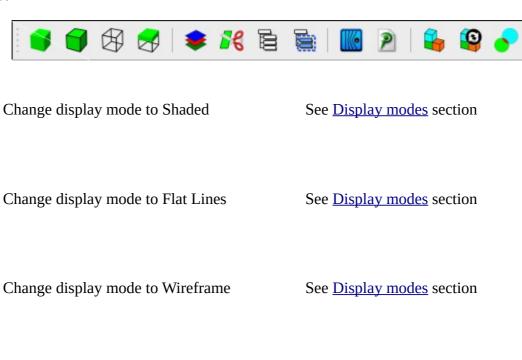
Menu bars are not available.

TOOLBARS

Three toolbars are available in PCB workbench:

- 1. PCB View.
- 2. PCB Settings.
- 3. Sketcher.

PCB View toolbar





Layers settings

Change display mode to Internal View

See <u>Layers</u> section

See Display modes section



Cut to board outline

See **Cut** to board outline section





Ungroup models in 'Parts' folder

See **Grouping parts** section



Group models in 'Parts' folder

See **Grouping parts** section



3D rendering: export to Kerkythea

See **Kerkythea** section



Export object to POV-ray

See Export object to POV-Ray section



Load file as assembly

See **Add assembly** section



Update selected assemblies

See <u>Update assembly</u> section



Collision detection

See Collision detection section

PCB Settings toolbar





Export PCB

See **Export board** section



Export BOM

See Export Bill Of Materials (BOM) section



Export hole locations

See Export hole locations section

Export hole locations report

Create drilling map

See <u>Create drilling map</u> section

See Export hole locations report section

Create drill center

See <u>Create drill center</u> section



Create new project

See <u>Create new project</u> section



Create PCB

See Create PCB section



Create glue path

See Create glue path section



Add annotation

See Add annotation section



Assign models

See <u>Assign models</u> section



Add model

See Add model section





Update models

See <u>Update models</u> section



Download models

See **Download models** section



Explode

See **Explode** section



Create constraint area

See <u>Create constraint area</u> section

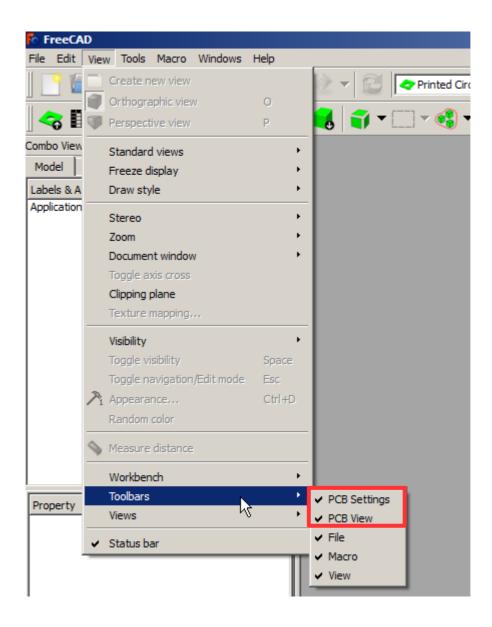


Bounding box

See **Bounding box** section

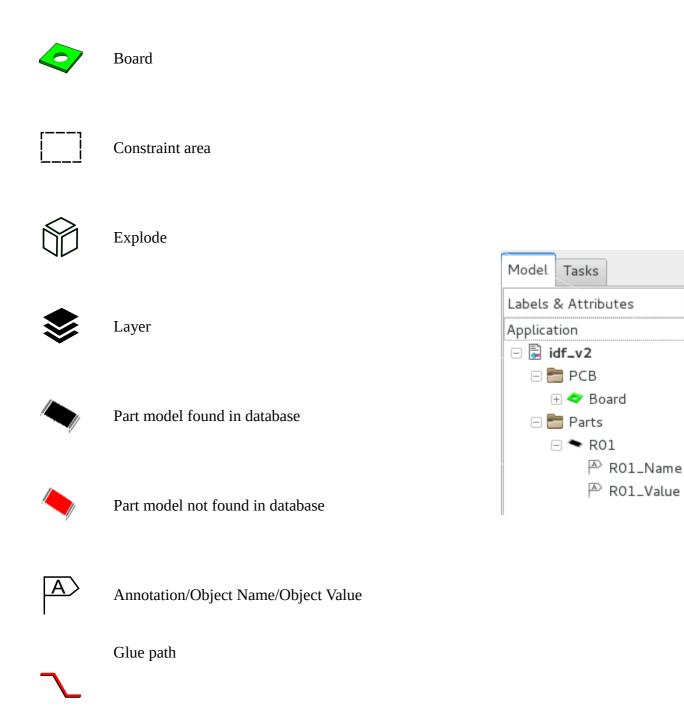
Displaying toolbars

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



SPECIFICATION TREE

There are few object types directly connected with PCB workbench. They can be identified in the 'Combo view' by the specific icons.





Main assembly object



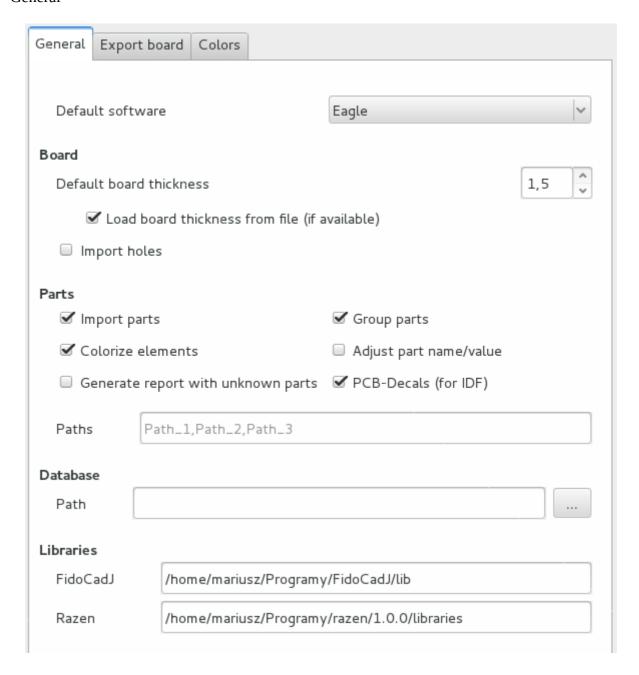
Main assembly subcomponent

More info about mentioned objects You can find in Objects properties section.

CUSTOMIZING WORKBENCH

To access to the PCB workbench settings You need to choose from top menu Edit → Preferences: section PCB. Preference tab for module contain three groups:

1. General



This section contains default settings for import process:

- Default software: this field allow You to set default used by you software,
- Board thickness: default value is 1.5mm,
- Paths to: database, 3D models, extra libraries,
- Checkboxes associated with importing parts/colors/holes
- Checkbox associated with generating report with unknown parts.

If checkbox 'Group parts' is checked, imported parts will be splitted to groups according to Category they belong.



It is recommended to keep parts and database.cfg file outside PCB folder.



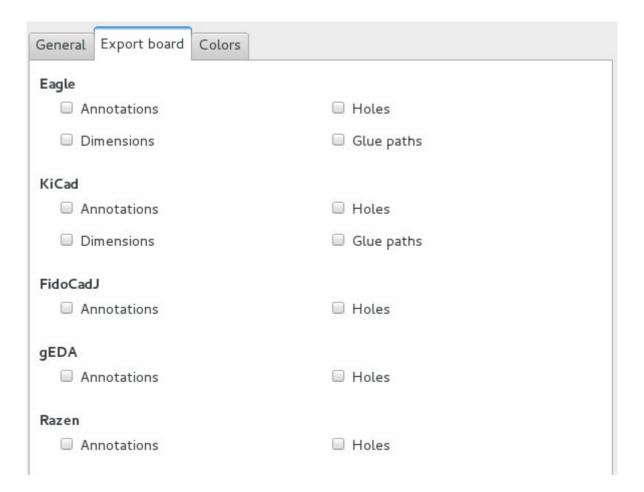
For more information about grouping parts see 'Grouping parts' section.



To set libraries for FidoCadJ you can indicate folder or main jar file.

2. Export board

Default settings associated with exporting board to one of supported formats can be set in this tab.



3. Colors

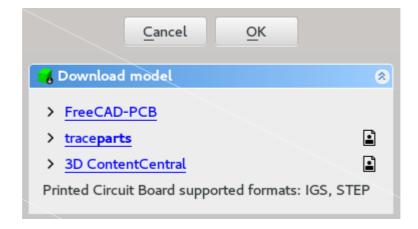
All default color can be set in 'Colors' section.



DOWNLOAD MODELS



Download model window appear in Task tab. Mentioned window contains links to sites when You can download for free 3D models.



Icons definition:



Registration is necessary to download models

Printed Circuit Board workbench supported formats:

- IGS,
- STEP at this moment colours are supported only for step files saved/created in FreeCAD. So to avoid problems with colours create part in FreeCAD or open an existing component in FC (made in another program) and save it as step with colours.

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.



It is recommended to keep parts outside PCB folder - to avoid data lost.



Default path is set on folder '/parts', which is placed in main PCB workbench folder.

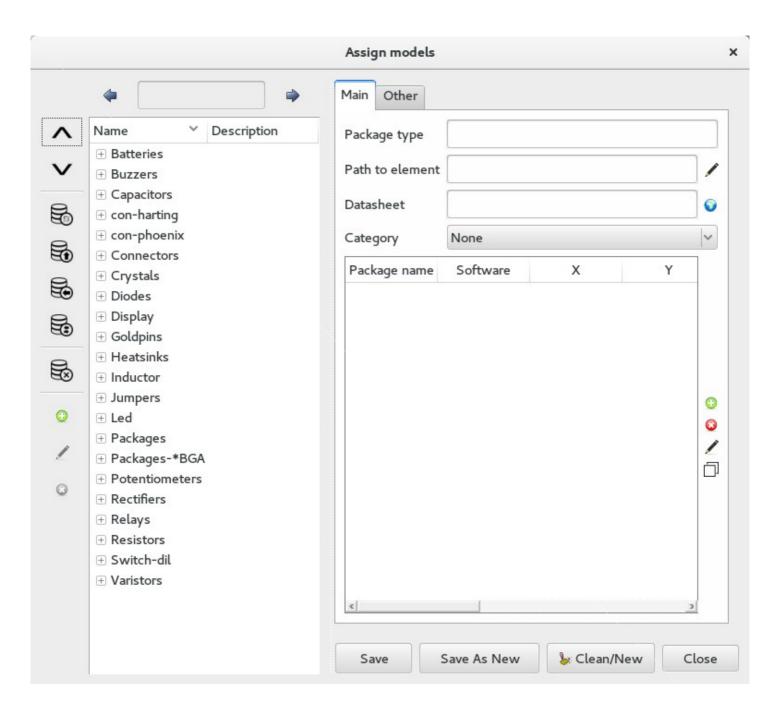


To add/remove paths you need to open Preferences window. More info you can find in section <u>Customizing workbench</u>.

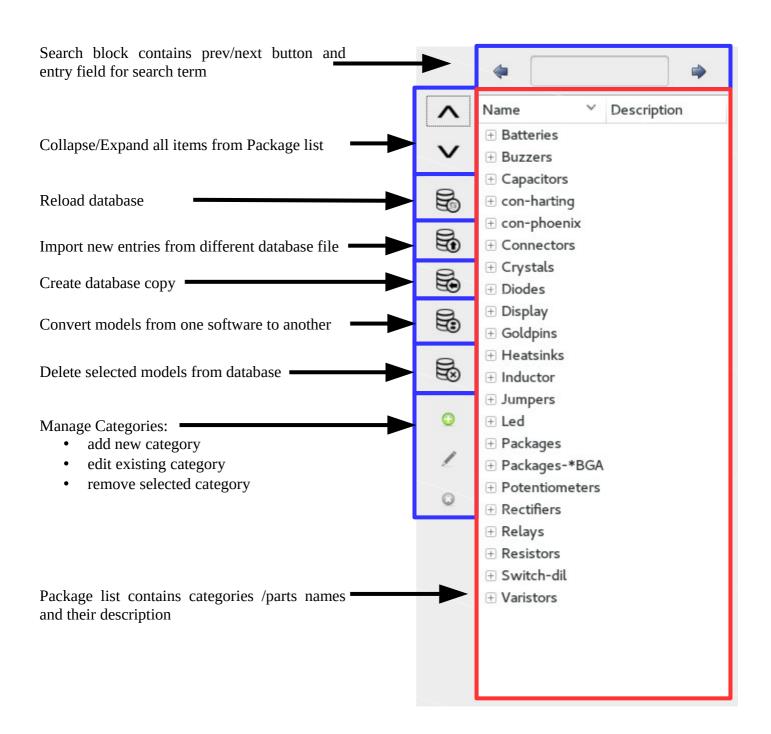
ASSIGN MODELS



Window 'Assign models' allow for assigning 3D models to corresponding part from one of supported software.



Window comprises two main columns. Left column comprises function necessary to manage parts in database. While right side comprises form where you can set (or edit) package data.



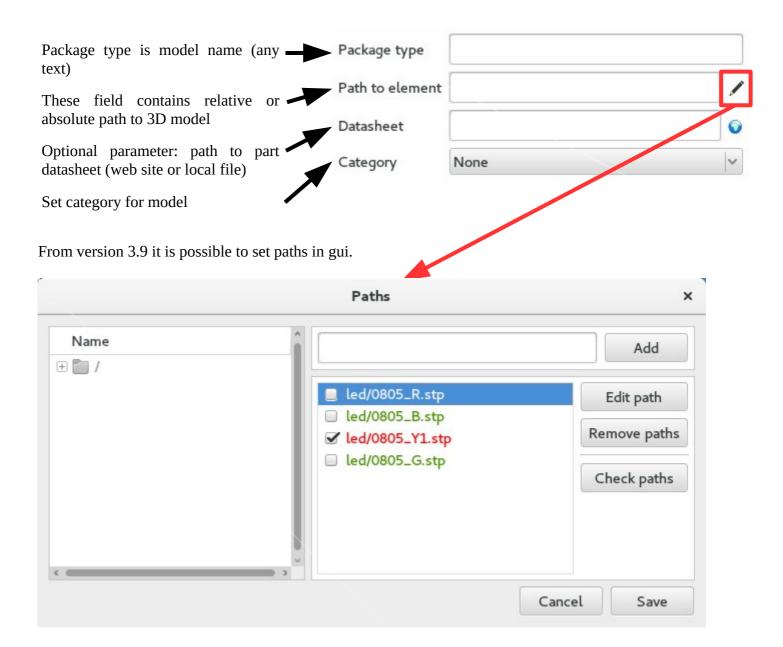


Categories are stored directly in FreeCAD. Tools \rightarrow Edit parameters => Preferences \rightarrow Mod \rightarrow PCB and variable partsCategories



After deleting model from database it is not possible to undo this operation!

Basics setting



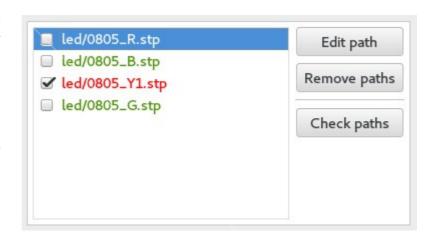
To add new 3D model (relative/absolute path) just choose model from left list or type directly in specified field and click Add button.



During initialization special function will check if saved before paths are still existing.

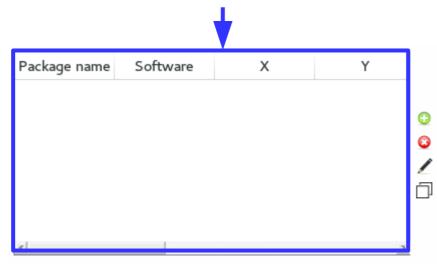
Green color means that the path is correct, while red means that something is wrong.

To check new paths you need to click button 'Check paths'.



Adding equivalent package for 3D model

First sections contain all package parameters, from one of supported Buttons from right side will helps You in softwares, necessary to load specific 3D model.



adding new /deleting or editing entries.

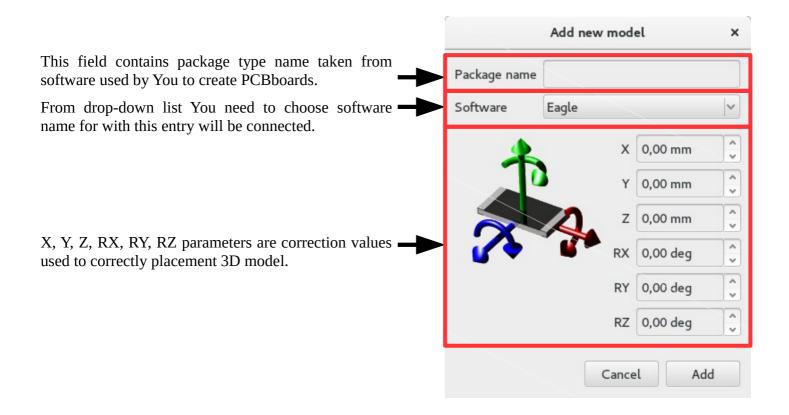
After clicking button 'Add' special window will appear, that allow You to set parameters for new package.

After clicking 'Edit' button will appear window, that will contain all settings for current selected model.

Clicking '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.

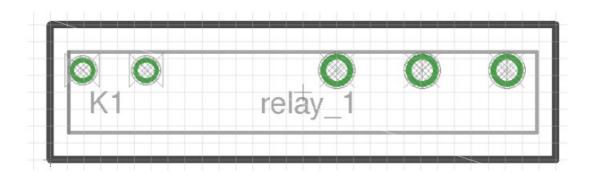


Adjust part name/value

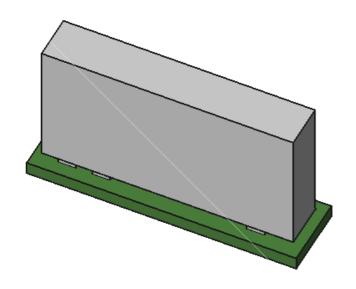
Option 'Adjust part name/value' allows to automatic placing objects name/value in specific position.

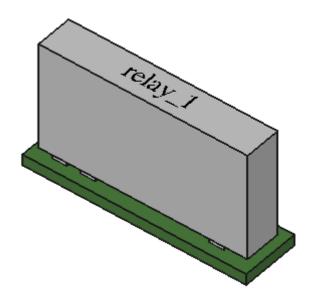
Parameter	Visible	j	Х	Υ	Z	Size	Color	Align	
Name	True	>	0,00mm	0,00mm 葉	0,00mm	1,27mm		center	v
Value	True	>	0,00mm 井	0,00mm 🕂	0,00mm	1,27mm	1	center	v

Example:



	Parameter	Visible	Х	Υ	Z	Size	Color	Align
\checkmark	Name	True	0,00mm 🗦	0,00mm 葉	14,95mm 🕂	3,00mm 🗦		center
	Value	True	0,00mm 븣	0,00mm 🕂	14,95mm	3,00mm 븣		center





Result without option 'Adjust part name/value'

Result with option 'Adjust part name/value'

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.



Set model as socket

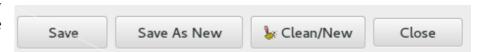
To set model as socket just mark checkbox for 'Set as socket' sign. In spinbox specify socket height.



Footer from right column contain buttons to saving form entries to database.

Save button will save form as new entry in database or will update package parameters (in edit mode).

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

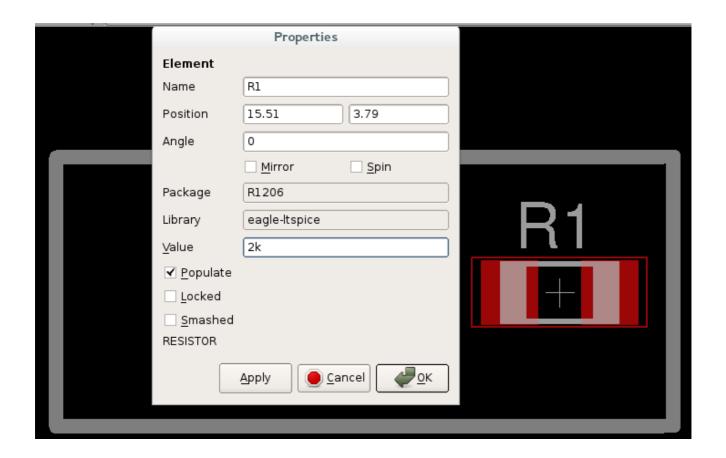


'Clean/New' button clean form.

Close button will appear only for GNU/Linux users.

Example of adding part to database

In Eagle we prepared board containing one part: R1 SMD resistor 2k in package 1206.



Open 'Assing models' window and:

- 1. In 'Package type' field write resistor_smd_1206. Package name is free text.
- 2. In 'Path to element' write resistors/R1206.

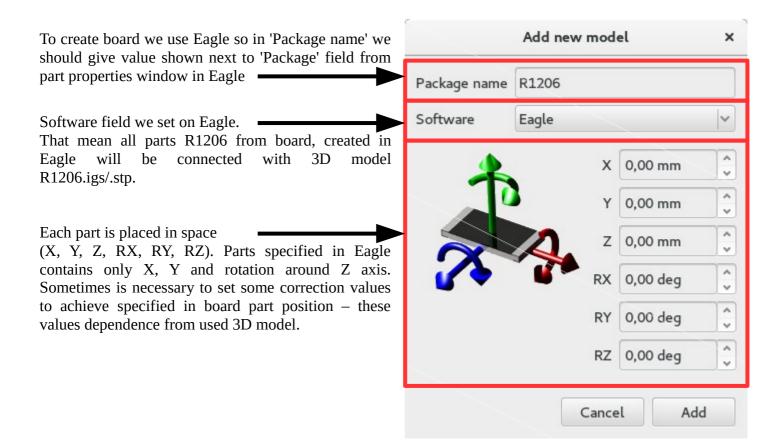


Model extension. resistors/R1206 mean that script will search at first file R1206.igs and then R1206.stp, so course path to element can also ook like resistors/R1206.igs or resistors/R1206.stp.



resistors/R1206 – script will search model in one of relative paths set in configurations.

- 3. Datasheet field may remain empty.
- 4. Package category may remain as 'None'.
- 5. Next step explains how to add new 'connection' between 3D model and part added on board.



- 6. Click add button.
- 7. This model is not socket and doesn't contain it, so we will skip specific checkboxes.

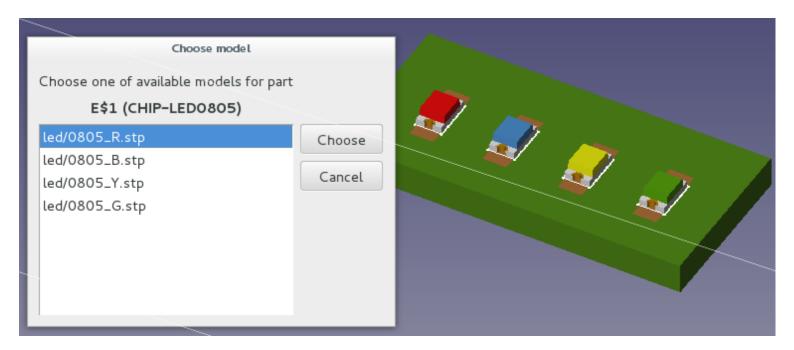
- 8. To save form in database click button 'Save'.
- 9. Thats all, now just reload board or update 3D models for R1206 packages.

Multi model definition for one part

There is a possibility to set more than one 3D model for one package. To do this just split path to all models by character ';' in field 'Path to element'.

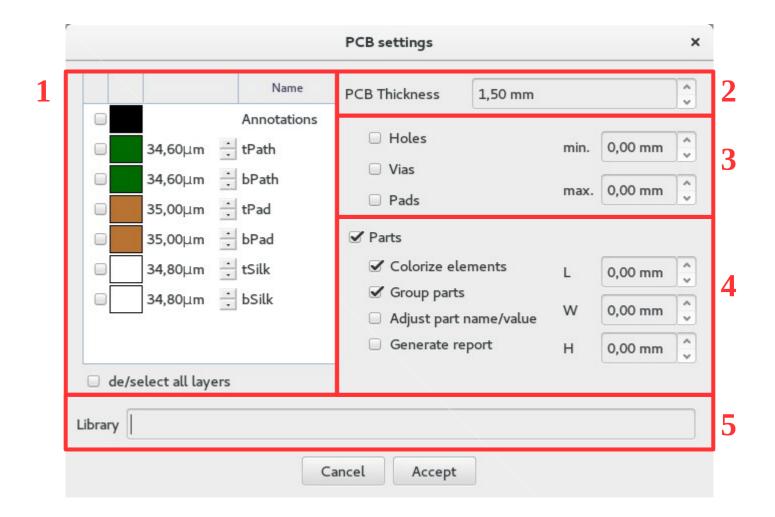


This function is useful for parts which only different is color – the same correction values are set for all models. For packages where we set multi models, special window will appear during board loading or parts updating.

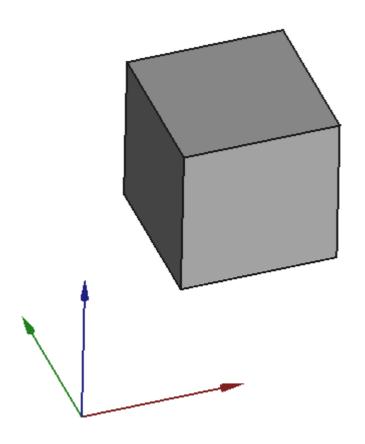


OPEN/IMPORT BOARD

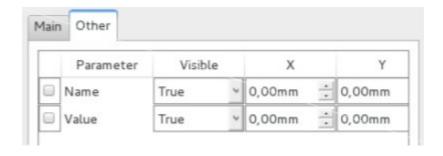
During open/import process special window will appear, in which we can set basic parameters of the board.



- 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.



Option 'Adjust part name/value' allows to automatic placing objects name/value in specific position. This option is connected with functions from 'Assign window'.



5. In some cases it is necessary to define path to specific program libraries.

Unit system

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

Below You can find information, which part of imported files are supported by Printed Circuit Board workbench.

	РСВ											
Soft name		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											
ZenitPCB	zpc											
TARGET 3001!	t3001											
FidoCadJ	fcd											
SCOOTER												
Razen	rzp											
IDF v2	idf											
IDF v3	idf											
IDF v4												
TurboPcb	apcb											
DipTrace	asc											
HyperLynx	HYP											

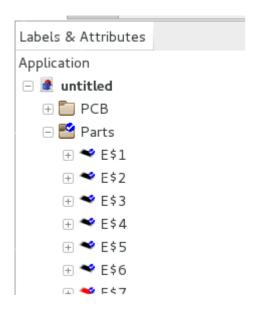


GROUPING PARTS

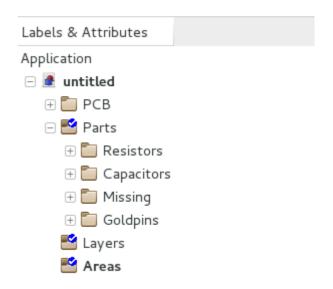




These options allow you to group/ungroup parts, according to Category they belong.



Ungrouped parts



Grouped parts

These options are also available in:

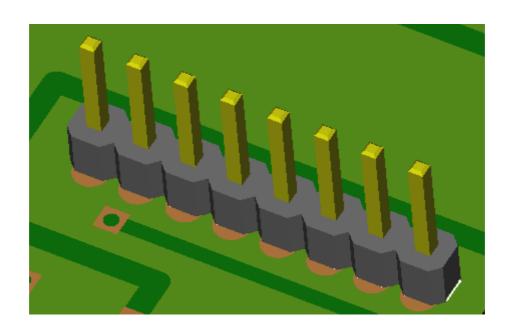
- · open/import window,
- update parts window,
- add new model window.

DISPLAY MODES

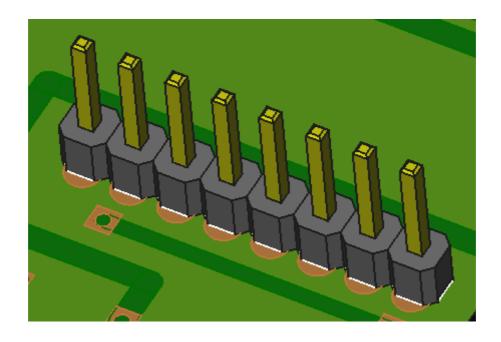
Display modes function can quickly and in easy way change display representation of shapes in project.

Available types:

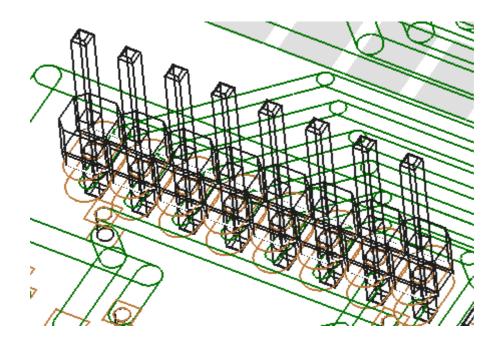
1. Shaded: border lines are hidden.



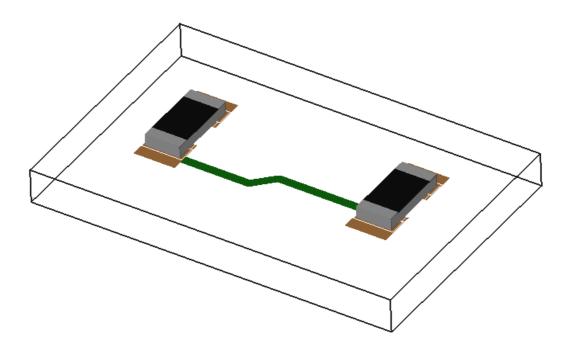
2. Flat lines: surfaces and border lines are displayed in one time.



3. Wireframe: only border lines are displayed.



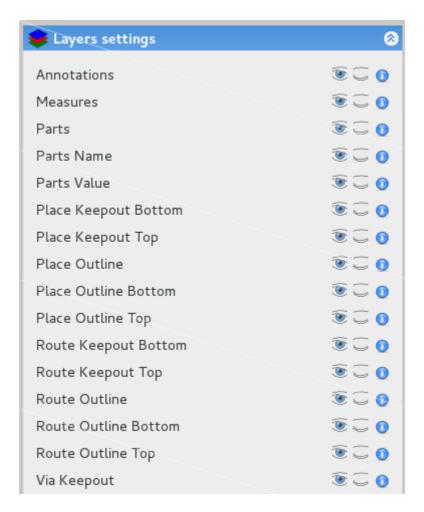
4. Internal View: for board only border lines are displayed, rest is displayed in Flat lines mode



LAYERS



Layers settings windows help in managing of currently displayed board layers. Layers settings window appear in 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.

Python

To manage layers by Python You need to import PCBlayers.

This module contain few functions:

1. display(TypeId): display layer with a given ID

TypeId: specific list

2. blank(TypeId): blank layer with a given ID

TypeId: specific list

3. toggle(TypeId): toggle visibility state of layer with a given ID

TypeId: specific list

4. state(TypeId): plirt state of layer with a given ID

TypeId: specific list

5. list(): return list of all available layers

'layerName: {'info': 'info', 'id': TypeId}

6. listTXT(): print all available layers

Example:

```
import PCBlayers as layer
```

```
lay = layer.list()
```

lay = lay["Annotations"]['id']

layer.blank(lay)

layer.state(lay) # False

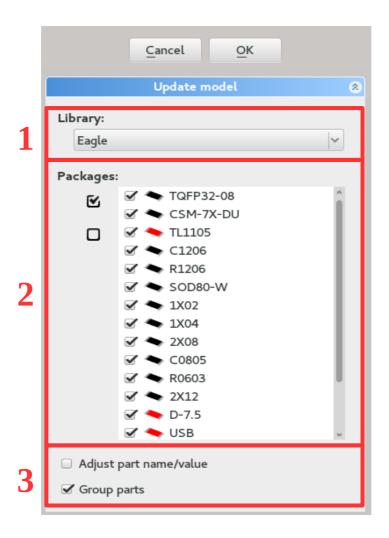
layer.toggle(lay)

layer.state(lay) # True

UPDATE 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.
 - Group parts: grouping parts in tree according to Categories.



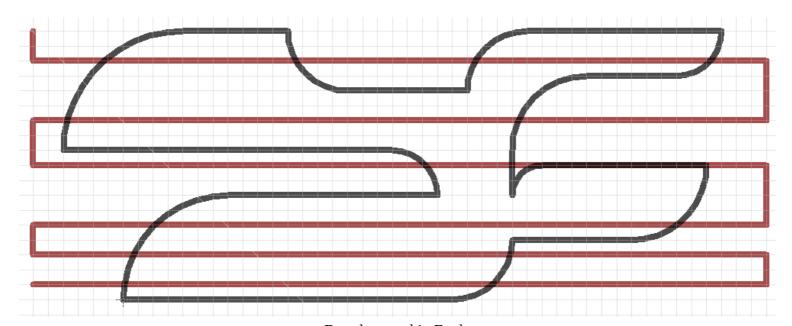
When selected component does not appear in specified library, model will be not updated.

CUT TO BOARD OUTLINE

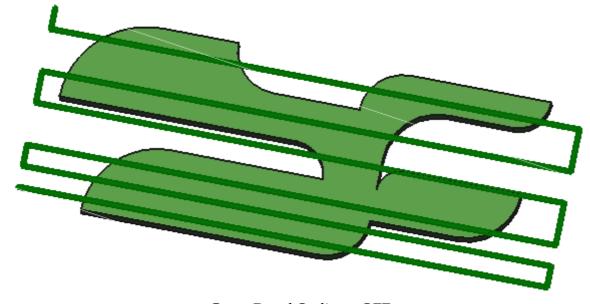


Sometimes it is necessery to display board like it will look after manufacturing. To do this just use option 'Cut to Board Outline'. Function will automatically blank/display all layers/paths that are outside of the board.

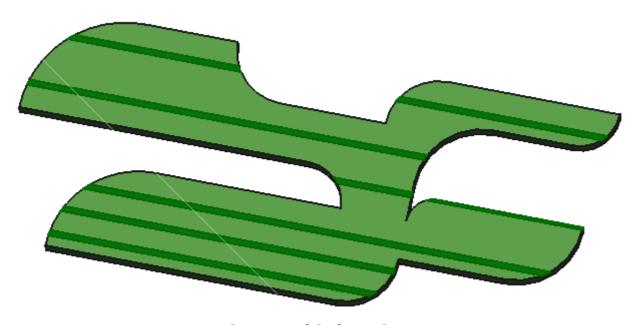
Example



Board created in Eagle



Cut to Board Outline = OFF



Cut to Board Outline = ON

EXPLODE



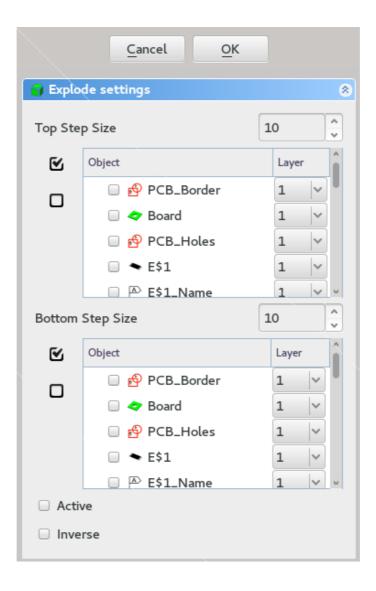
Explode function allows for assemblies to be quickly disassembled and shown in an exploded view.

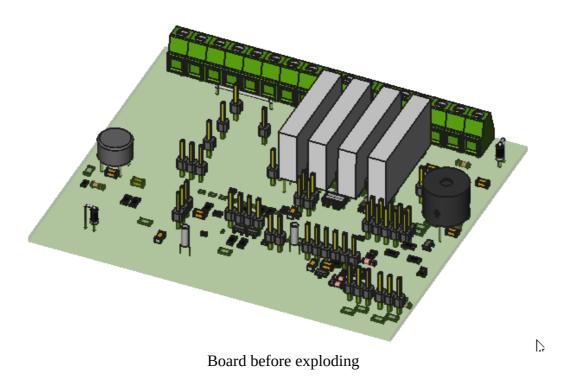
Explode contains two similar functions:

- 1. Fast explode: automatically explode all printed circuit board parts.
- 2. Explode: user selects which parts and how will be relocated.

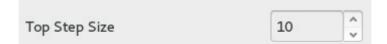
This task shows you how to manually create a explode object.

- 1. Load PCB and click Explode button
- 2.



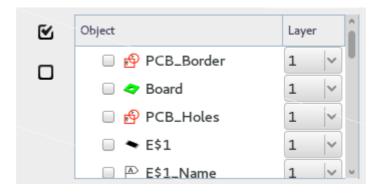


3. Set Top step size



Top step size determines distance between next levels.

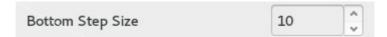
4. Choose parts – top side



Selected parts will automatically disappear from the bottom list.

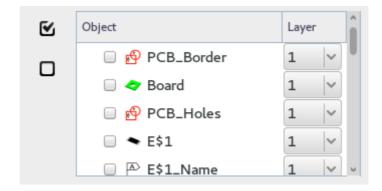
Layer parameter is a number, that will be multiplied by Top step size to determine new Z position for part after explode

5. Set Bottom step size



Bottom step size determines distance between next levels.

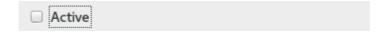
6. Choose parts – bottom side



Selected parts will automatically disappear from the top list.

Layer parameter is a number, that will be multiplied by Bottom step size to determine new Z position for part after explode.

7. Active



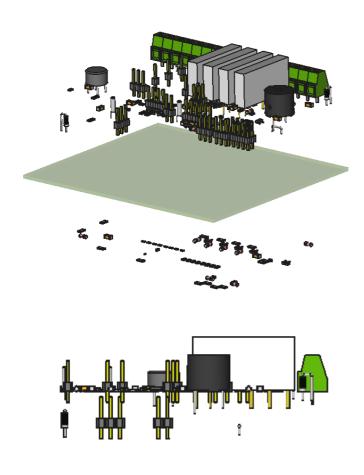
If this option will be checked explode object will be activated automatically after creation.

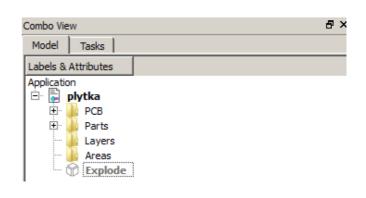
8. Inverse



To reverse top and bottom side check this option.

9. Click Ok button to finish









Fast explode function automatically splits parts between top and bottom layers.

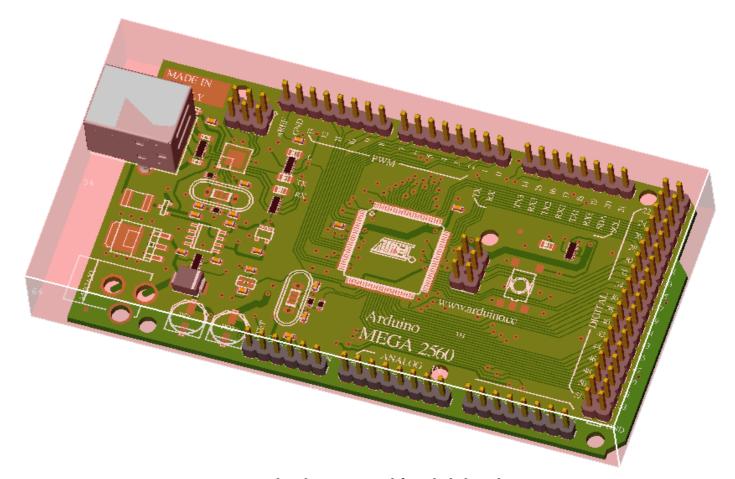
BOUNDING BOX



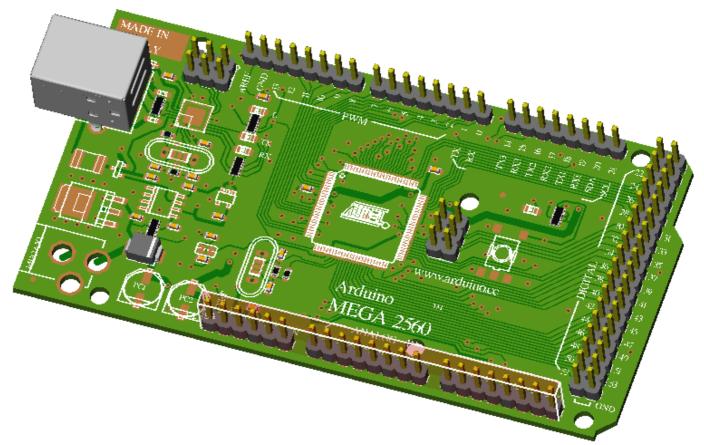
Bounding box is a smallest cuboid completely surrounds the object.

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 on them in FreeCAD.



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

Python

To create bounding box by Python You need to import PCBboundingBox

This module contain few functions:

- 1. boundingBox(): create bounding box for whole board.
- 2. boundingBoxFromSelection(): create bounding box for selected objects.

Example 1:

import PCBboundingBox as box
box.boundingBox()

Example 2:

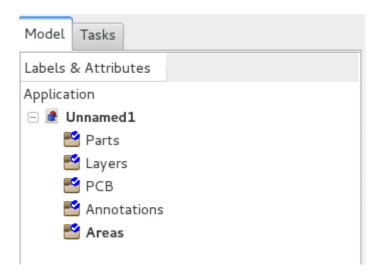
import PCBboundingBox as box

- a = box.boundingBox()
- a.Length # 179.546 mm
- a.Width # 104.27 mm
- a.Height # 2.155 mm

CREATE NEW PROJECT



Before starting project it is recommended to press Create new project button. It will create necessary groups in Specification Tree used later by script.



Meaning of groups:

- Areas: group will store constraint areas,
- PCB: group will store board object,
- Layers: group will store layers,
- Parts: group will store all parts used in project,
- Annotations: group will store annotations.



It is recommended to press Create new project button before starting project, but it is not necessary – script will check if specific group exist and, if desired will create it.

Python

To manage groups by Python You need to import PCBgroups module.

This module contain few functions:

- 1. createGroup_Parts(): create Parts group.
- 2. createGroup_Layers(): create Layers group.
- 3. createGroup_Annotations(): create Annotations group.
- 4. createGroup_Areas(): create Areas group.
- 5. createGroup_PCB(): create PCB group.
- 6. setProject(): create all available groups.

Example:

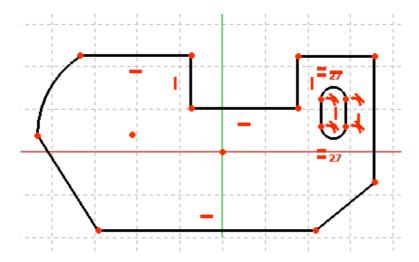
```
import PCBgroups as group
pcbG = group.createGroup_PCB()
pcbG.Label() # u'PCB'
```

CREATE PCB

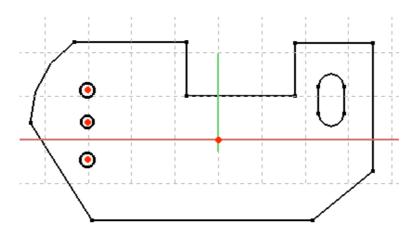


This task explains how to create a board and design its geometry from scratch.

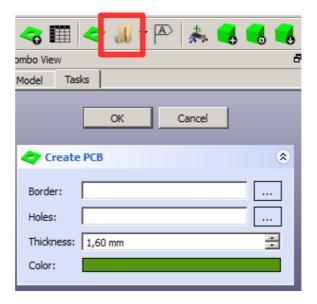
1. Create Sketcher with contour of fthe boaard



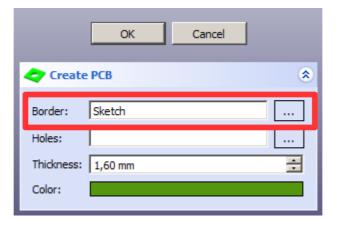
2. Create new Sketcher with holes



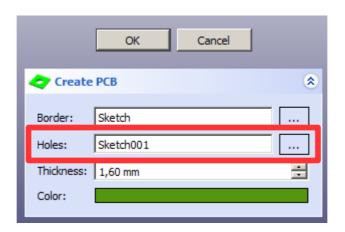
3. Click the **Create PCB** button



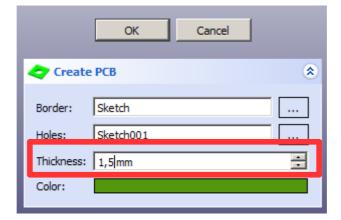
4. As **Border** point to Sketcher with contour of fthe boaard



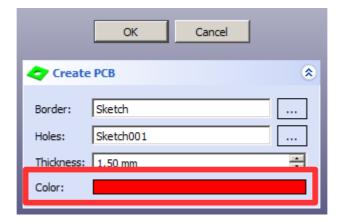
5. As **Holes** point to Sketcher with holes



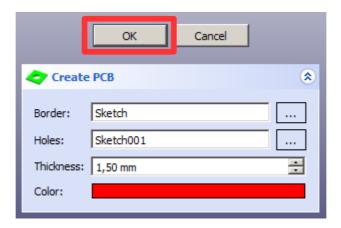
6. Set board **Thickness**



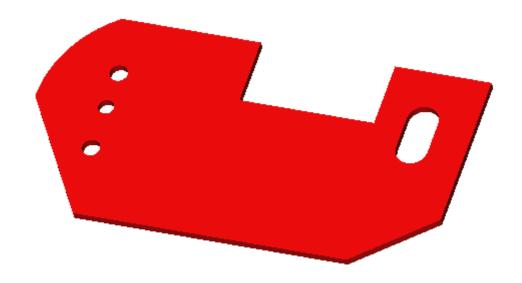
7. Set board Color



8. Click **OK** button to finish



The board should be generated according to settings. The board has been created from scratch.





Only one board can be generated per project



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



It is not possible to create board from a previously created part

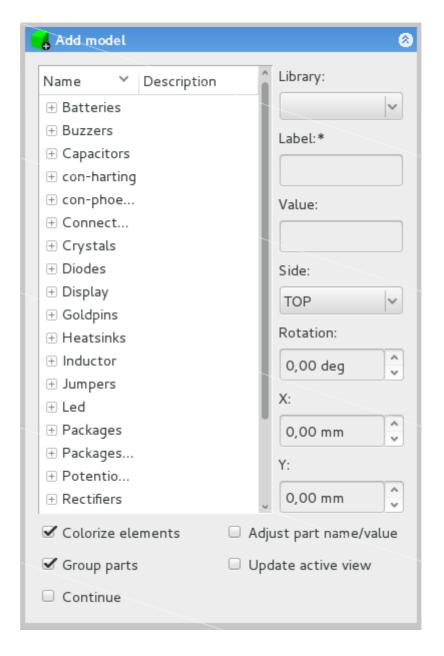
ADD MODELS



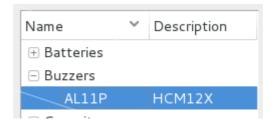
This task explains how to add new component to existing board.

1. Click button **Add model**





2. Select package – model type



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



4. Write component name



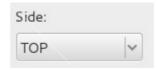


This field is mandatory

5. Add value

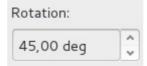


6. Choose side for new component on board



Component will be automatically rotated according to chosen side.

7. Set rotation value (rotation around Z axis)

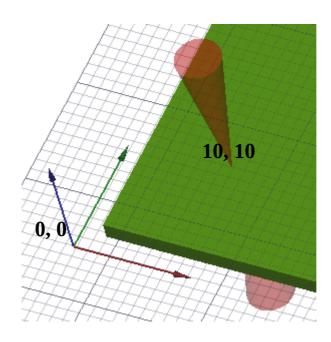


Value is in degrees.

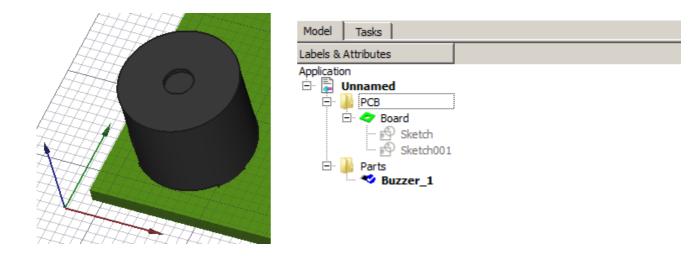
8. Set values for X, Y coordinates



Distance is measured from 0, 0 in 3D window. Actual position (model center) is representing in 3D view by red 'arrow'

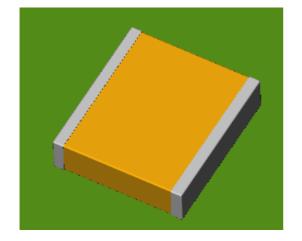


9. Click Ok button to finish

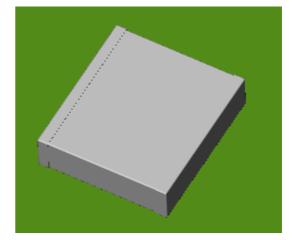


Add model tab contain five configuration options:

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.
- 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.
- 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.

ADD ANNOTATIONS

CREATE GLUE PATHS

CREATE CONSTRAINTS AREAS

EXPORT BOARD

EXPORT BILL OF MATERIALS (BOM)

EXPORT HOLE LOCATIONS

EXPORT HOLE LOCATIONS REPORT

CREATE DRILLING MAP

CREATE DRILL CENTER

ADD ASSEMBLY

UPDATE ASSEMBLY

COLLISION DETECTION

EXPORT TO KERKYTHEA

EXPORT OBJECT TO POV-RAY

OBJECTS PROPERTIES

FILE FORMAT

SCRIPTS

ERRORS CODE

LICENCE

CHANGELOG

TODO LIST

ERRORS