

Translation of: Eagle building block library by Commend International

The board (leiterplatte) , trace (platine) and the placement of new building blocks must be considered in order to come up with a sound mechanical design.

Technical Background:

In order to import mechanical dimensions of a building block from ECAD-system into MCAD, standards were created in 1989:

IDF: Intermediate Data Format V1.0

The latest version: V4.0

Generating IDF files is supported by all professional ECAD systems (Mentor, Zuken, Targe, PADS, PCAD,...)

Milestones of IDF Standards:

Find standards under: `\\Eacoco\Projecte\CAD EEAGLE\DOKU eagle\`

Solidworks 2006 SPx supports currently only IDF standard V3.0, meaning, the build blocs are limited:

1 high information with 1 closed outside contour.

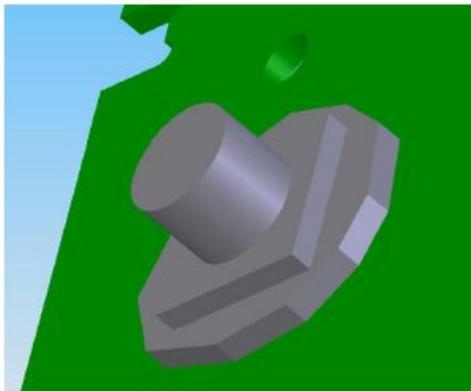
For an accurate design, build blocks data with only 1 contour and the max. height is sufficient.

Creation of an EAGLE library:

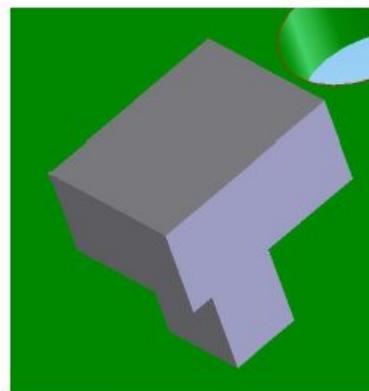
1) Building blocks:

A building block can be assembled out of as many height informations as you want. The various segment heights can be assembled like Pyramids, however a "standing on its tip" Pyramid is not possible.

Expl:



graph 1a) POSSIBLE



graph 1b): NOT POSSIBLE

A closed contour is of each height information is PARAMOUNT in order to extrude it.

Building block contour (R, L, C, IC, ...) is placed on Layer 57 (tCAD) and Layer 58 (bCAD).

Layer 57: planview of the building block (standard)

Layer 58: penetration of a building block through a trace (platine) i.e mechanical mounting, place on Layer 58

All building blocks are normally, exclusively drawn on Layer 57. View exceptions are connectors with mechanical mounting, screws, solderable building block legs, which penetrate the print, protruding the bottom side.

Line thickness of the contour relates to the building block height:

Building block height: $\text{Line thickness} = \text{building block height [mm]} / 1000$ (1mm = 1000mic)
 (expl.: Building block height 12.6mm is drawn as 12.6 mic.)

Necessary milling (cut out), holes in building block -> line thickness = 0.0 [mic]

Expl.:

see pic.

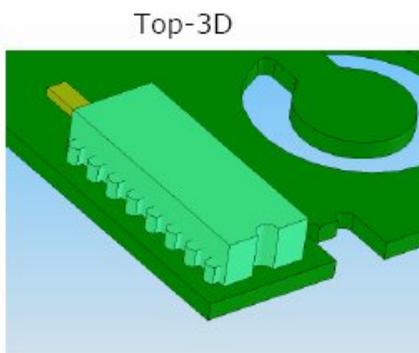
- 1) Line thickness 8 mic = cylinder 8mm height
 - 2) Line thickness 3 mic = rectangular 3mm height
 - 3) Line thickness 1.6mic = polygon 1.6mm
- ➔ **result see graph 1a**

Drills/holes do not have to be treated specially -> automatic recognition and incorporated into the board.

Expl.:

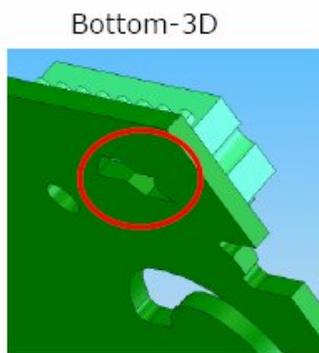
- 1) Connector-building block 3.7mm -> line thickness = 3.7 [mic]
 Connector-cut out for mechanical mounting: Polygon with 0.0 [mic]
 Connector-hole as positioning support: hole diameter 2.1 [mm]

- 2) cut out beneath IC (see graph 2c (1))



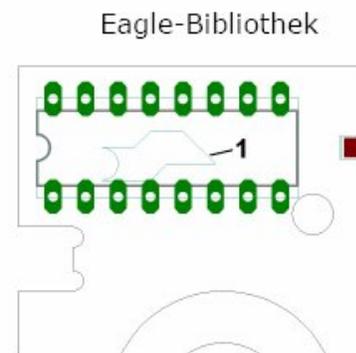
Graphik 2a

graph 2a (Top 3D)



Graphik 2b

graph 2b (Bottom 3D)



Graphik2c

graph 2c (Eagle library)

- 2) Empty board:

Contour of empty boards on Layer 50

A closed contour is PARAMOUNT in order to extrude it later

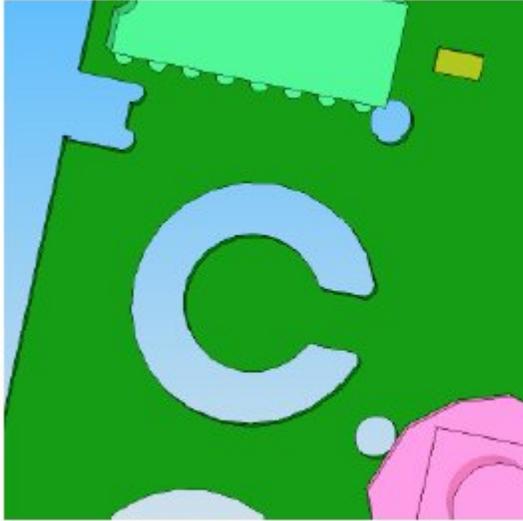
Line thickness of outside contour is same as board thickness.

Trace (platine) height -> $\text{line thickness} = \text{building block height [mm]} / 1000$ (1mm = 1000mic)

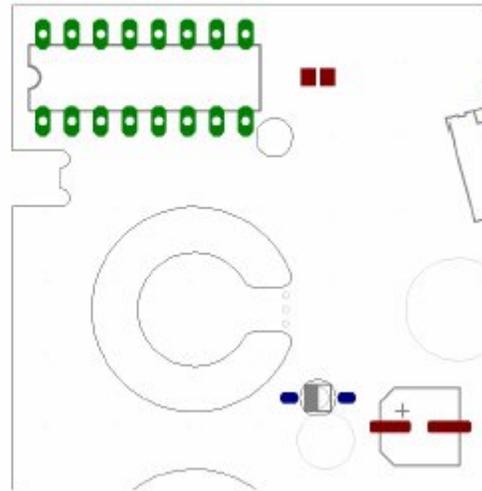
Necessary cut out, holes of trace (platine) -> line thickness = 0.0 [mic]

Expl.:

Line thickness board outside contour = 1.6 [mic], print cut out = 0.0 [mic]



graph 3a



graph 3b