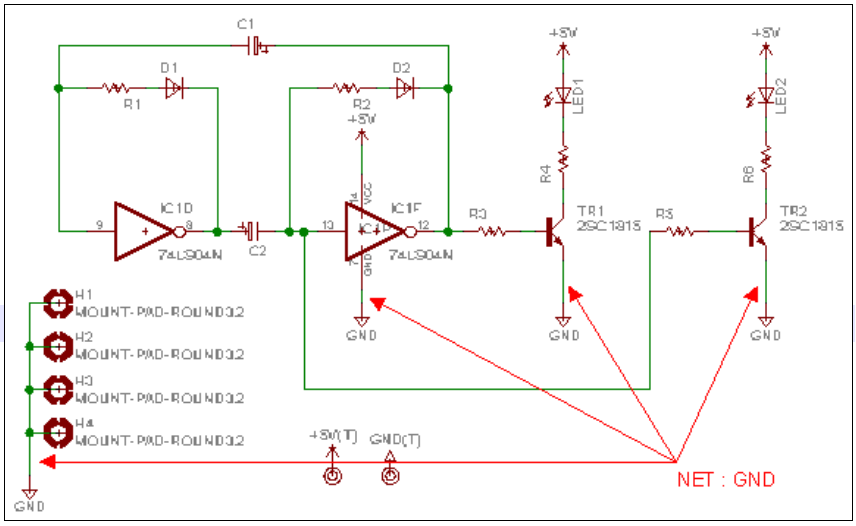
PCB Groundplanes in Eagle

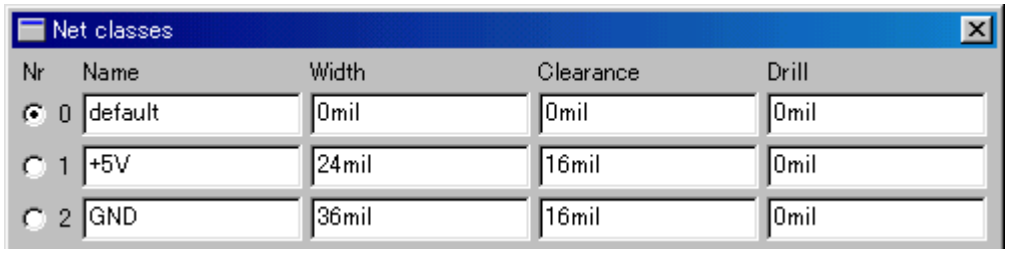
In case of making of a PCB, there is technique which makes a wide grounding pattern except the signal and power pattern. This technique is used for the circuit for the high frequency or when preventing an influence by the noise from outside and so on. You can also save much copper when making a printed board by milling. The polygon function is used to draw such a grounding pattern.



Schematic

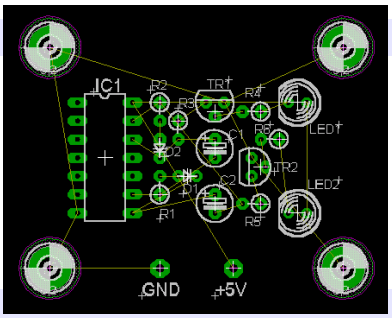
The circuit on the right is an a-stable multivibrator and will be used as an example throughout this tutorial.

"Groundplanes" are called that way because they're usually connected to the ground or mass but they can also be used for other connections such as positive and negative power supply, live and neutral AC connections, etc. All ground connections in the schematic above are interconnected by the grounding symbol and therefore the nets they're connected to are automatically named "GND". This name is used for the name of the polygon. So it's better to confirm the name of the grounding wire by the *Info* command after drawing a schematic.

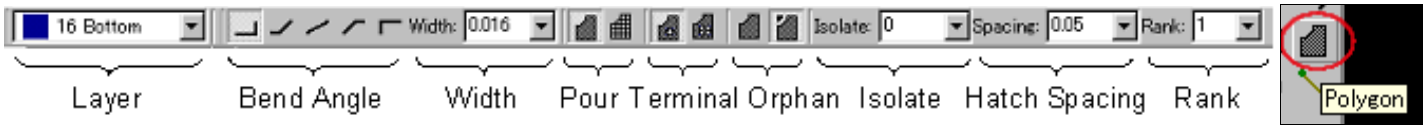


Circuit board

Clicking the *board* button once will generate an unrouted PCB. You can route the signals first and add the groundplane afterwards or the other way round but keep in mind the ERC check will fail as long as the groundplane hasn't been drawn yet.

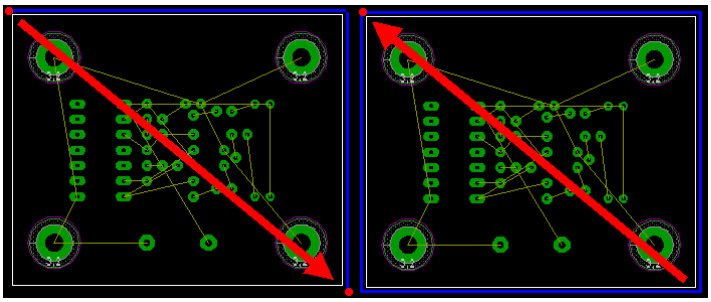


Polygon settings

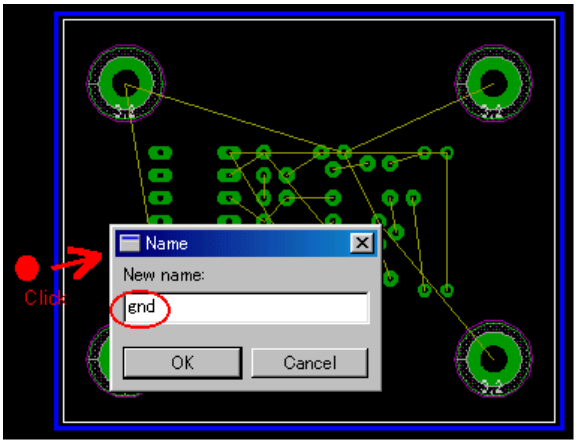


Layer	This is the layer specification of the polygon to draw. 1 is the component side (auburn) and 16 is the wiring side (dark blue).
Bend angle	This is the bend angle specification when making a polygon. From the left side: 0 : Starting point - horizontal - vertical - end 1 : Starting point - horizontal - 45° - end 2 : Starting point - end (straight connection) 3 : Starting point - 45° - horizontal - end 4 : Starting point - vertical - horizontal - end

Width	This is the specifying of the precision to draw a polygon. Usually, the width which is the same as the signal pattern is good. The setting unit is an inch.
Pour	This is a pouring pattern specification. Left is Solid. Right is Hatch.
Terminal	This is a shape specification for the terminal which is connected with the polygon. Left(off) is combining. Right(on) is showing.
Orphan	This is the specification of the blank area of the polygon. Left(off) is no making up with polygon. Right(on) is making up with polygon.
Isolate	This is the interval specification between the signal pattern and the polygon. When this set value is smaller than the value specifying by DRC(Design Rule Check) or Net Class, either of DRC or Net Class a big value is used. The default is 0. The setting unit is an inch.
Spacing	When "Hatch" is selected by the item of pour, this is the interval specifying of the hatch. The setting unit is an inch.
Rank	This is the priority specifying of a polygon. It is effective when more than one polygon is piled up. When the figure is small, the priority is high. "1" is a top priority.



Draw a polygon for the grounding in the periphery of the PCB area. Click the outside of the upper left of the board to set a start point. Next, click the outside of the lower right to set a terminal. It isn't necessary to drag the cursor. The red mark shows a click point. Move a pointer to the start point at the upper left and click it. A polygon area is set in this.

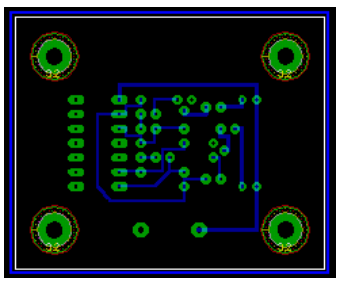


Set a polygon name. A polygon is connected with the grounding by this. After clicking the *Name* button, click the outside of the polygon and type the name of the grounding. A dialog box will be displayed after clicking the OK button. Eagle will connect the groundplane to the ground once the connection has been confirmed.

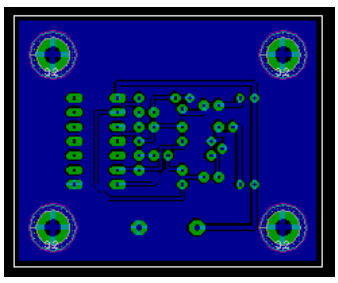
The groundplane is drawn and (re)rendered after clicking the *ratsnest* button. It's now in the condition where signal patterns aren't drawn yet. However, the polygon is redrawn excluding the signal paths if signal patterns are routed.

Change polygon display mode

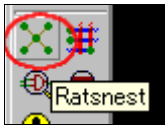
The display mode of the polygon has two types: An outline mode and a real mode. As for the outline mode, only the frame of the polygon is shown and the polygon isn't drawn in the board. This mode is convenient when changing a pattern and so on. As for the real mode, a polygon is drawn in the board. This mode is convenient to confirm the condition of the polygon.



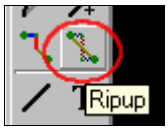
Outline mode



Real mode



The *ratsnest* tool switches from outline to real mode by re-rendering the PCB.



The *ripup* tool switches from real mode to outline mode. Activate this tool and click once outside of the PCB to execute the command.

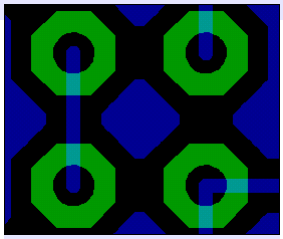
Only the data of the outline mode is recorded to the file of EAGLE. The displaying of a real mode is shown with the CAM processor from the data of the outlines. A CAM processor is started up by Ratsnest and real mode displaying is done. The outline mode is shown when restarting a project.

Deleting a polygon or groundplane

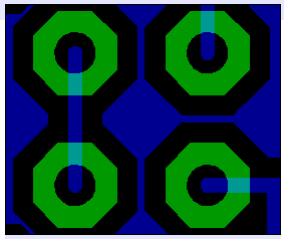
Switch over to real mode and remove the polygon by selecting the *remove* tool and clicking the border of the polygon. The polygon will change shape and size as long as some of the borders are still present. Redo this until the polygon is completely removed.

Width

The detail of drawing a polygon can be specified by the *width* parameter. In the manual of EAGLE, it is referred to with "Typically you should keep the polygon width in the same range as your other wires". When drawing a polygon with the hatch, *width* becomes the width of the line of the hatch.



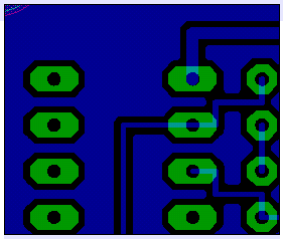
0,016 inch



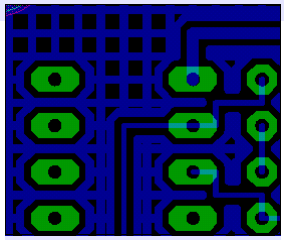
0 inch

Pour

The pattern of the polygon can be specified. Generally, it's *solid* but *hatch* specifies the intervals at which a line of the pattern is drawn to make the PCB lighter or just more 'attractive'.



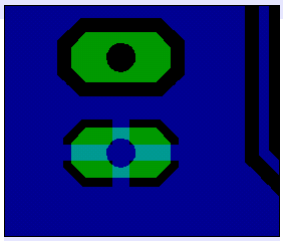
Solid



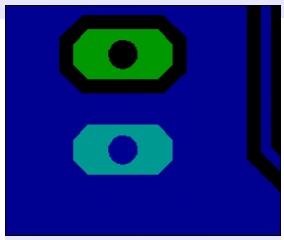
Hatch: 0.05mil

Ground pins

There are two kinds of ways to connect between the ground pins and the ground plane: Following the shape of the solder pad and including the solder pad.



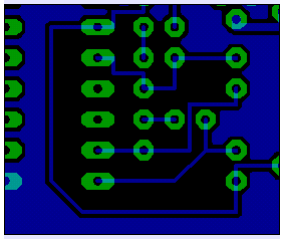
Following the shape



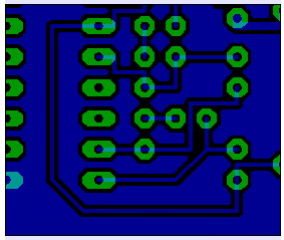
Including solder pad

Orphan

When surrounding a signal pattern by the ground pattern polygon, the area where it isn't possible to draw a ground pattern polygon sometimes exists. This setting defines whether Eagle should draw these unconnected polygons. These patterns sometimes influence circuit operation in high frequency circuits.



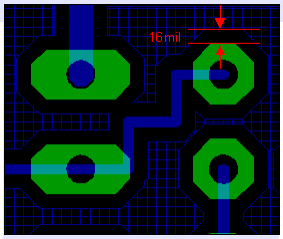
Disallowing orphans



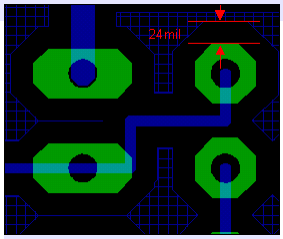
Allowing orphans

Isolate

The default value of Isolate is 0. In this case the interval between the signal pattern and the polygon is drawn with the bigger value either of DRC or NetClass. The set value will be applied where it exceeds the value specified by DRC or net class. The minimum clearance in DRC is 8 mil but 16 mil is set to the clearance of GND in Net Class.



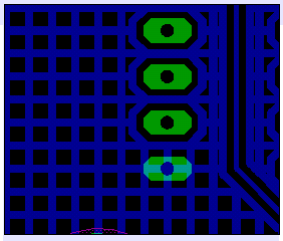
Isolate: 0 inches

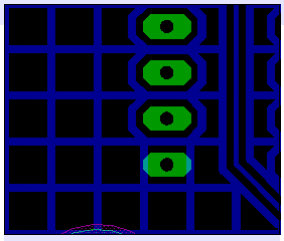


Isolate: 0,024 inches

Spacing

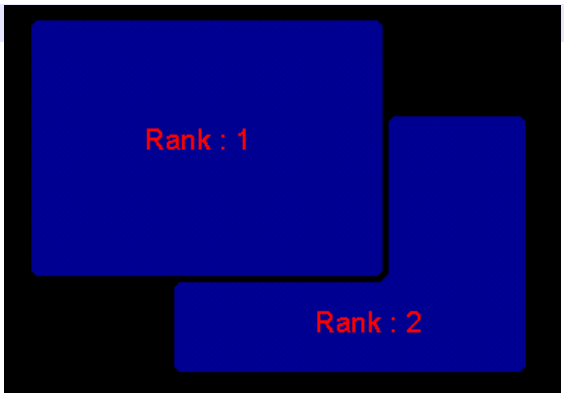
The hatch interval when drawing a polygon with the hatch is specified by this. It becomes the same as *solid* because the line of the hatch overlaps when *spacing* is narrower than *width*.


Spacing: 0,05 inches


Spacing: 0,1 inches

Rank

The smaller the value, the higher priority. When a polygon overlaps, the polygon with the higher priority is drawn.



Information provided by http://www.interq.or.jp