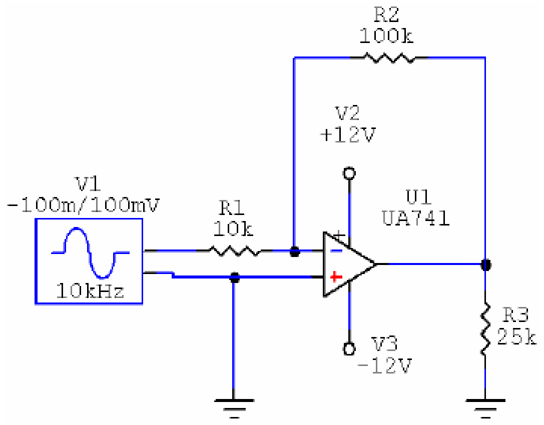
Ⅰ. Drawing PCB patterns with Easy CAD software

1． Drawing PCB patterns with Easy CAD

EasyCAD is a Computer Aided Design software to help you draw PCB patterns easily.
A pattern can be smoothly drawn if you prepare a sketch of the pattern based on the circuit diagram before you start drawing. Here, we will explain how to draw a PCB pattern from a circuit diagram of inverting amplifier.



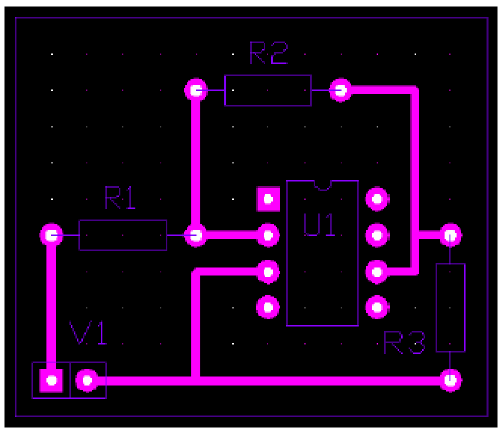


Fig. 1　Inverting amplifier　　　　　Fig. 2　Finished PCB pattern

1-1.　Initial Setting

To start EasyCad software, go to “Start” menu of Windows, then select “All Programs” / “Mits Design Pro (English)” / “Design Pro”.　Once DesignPro is started, select “EasyCAD” application from the “Application” Menu at the top right corner of the DesignPro screen.

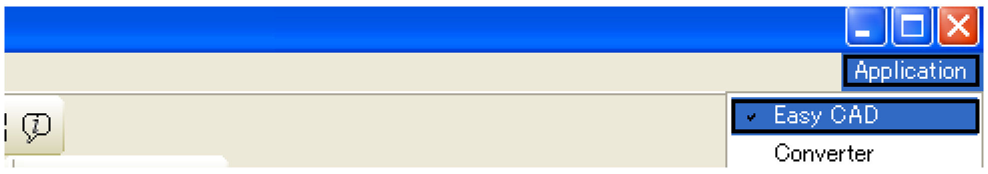


Fig. 3　Selecting Easy CAD

(1) Selecting Measuring Unit

You can select either “inch” or “mm” as a unit of measurement by clicking “Unit” pull down menu shown below. Here, we select “mm”.

* You can also select Inch as measuring unit.

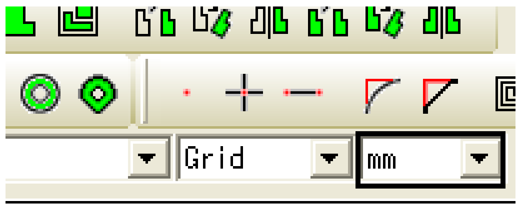


Fig. 4 Selecting measuring unit

(2) Displaying Grid

Grid is used as reference points so that you can easily arrange pads, parts, or patterns smoothly.

* Typically, grid spacing is set at 2.54 mm (0.1 inch) or pin to pin spacing of IC’s. If you need to run a pattern in between IC pins, set grid spacing at 1.27 mm or half of 0.1 inch.

* You can enter a numeric formula such as “2.54/2” instead of 1.27 as grid spacing.

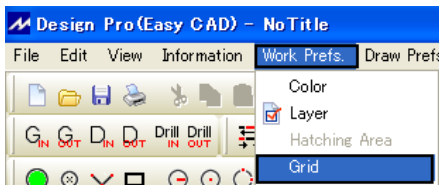


Fig. 5 Choosing Grid

〔Menu〕 / 〔Work Prefs〕 / 〔Grid〕

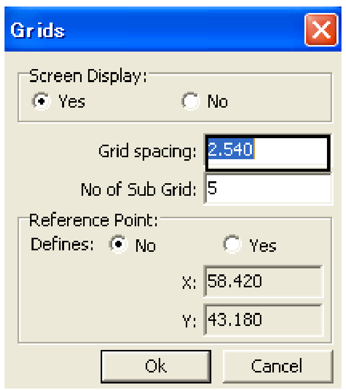


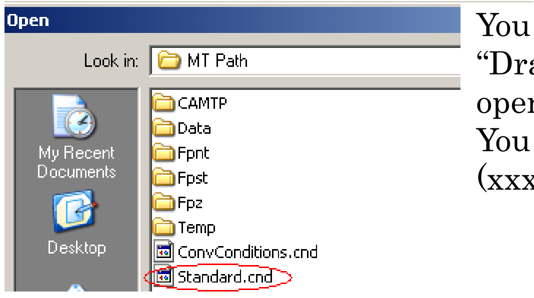
Fig. 6 Grid Spacing

Grid Spacing 2.540 (mm)

You can define a reference point of the grid by checking “Yes” to “Defines” sub menu of “Reference Point”.

If you select “Yes”, then press “OK” button, “Specify a reference point” shows up in comment box at the left bottom of the EasyCad screen. You can click any position in the screen to define a reference point.

(3) Loading a Template



You can load a template by selecting "Draw Prefs." / "Load Conditions", and open "standard.cnd".
You can also select your own template file (xxx.cnd) from this menu.

Fig. 7 Loading a Template

1-2. Drawing a PCB Pattern

In this section, let us explain how to draw a single sided pattern (bottom layer, or soldering layer) as shown here.

EasyCad display shows a top view from the parts side. The pattern on the bottom layer (soldering side) is shown as a transparent view from the top.

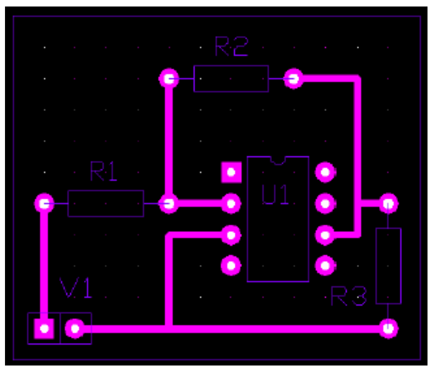
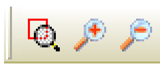


Fig. 8 A PCB Pattern

(1) Drawing board outline

Select "Subsidiary Top" layer to draw an outline of the board. This layer is used to draw reference points, diagrams and other information, and is not used for machinery work such as milling or drilling.

Choose appropriate display magnification factor using display tool .

Select "Subsidiary Top" from the layer pull down menu. 

Click  drawing tool to draw a rectangular by specifying two diagonal points.

* To draw a circle, use one of the circle drawing tools from .

To draw a polygon, use polygon tool .

1) When you click  button, the status bar at the bottom left corner of the drawing area says “specify a first corner”. Move the cursor to the first corner of the rectangle and click.

2) The status bar now says “specify a second corner”. Hit TAB key to show “Input Position” window. Enter “38,36”, then hit OK. An rectangle is drawn.

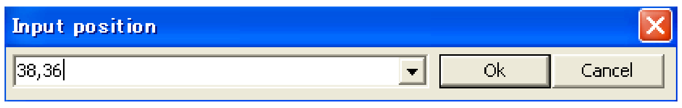


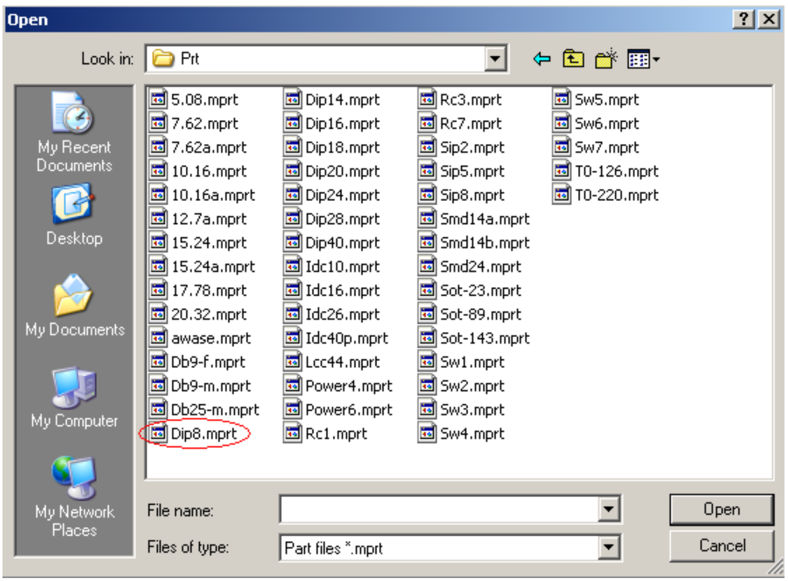
Fig. 9 A rectangle of 38 x 36 mm is drawn.

3) To end drawing an outline, hit cancel in “Input Position” dialogue, then hit ESC key on the keyboard.

(2) Arranging parts and pads

To arrange a part or a pad, you can either use a part from parts library or to arrange each pad one by one. We will explain how to edit prearranged pads in a parts library file in the following pages.

1) Selecting a part layout from "parts library"



Please refer to "EasyCADPartsLibrary.pdf" in "PDF File" folder of the CD manual for the detailed explanation of parts library.

Fig. 13 Opening "Dip8.mprt" file

(a) From the menu bar, select "File" / "Place Part", select "Dip8.mprt(Dip8pin)" file, then hit OPEN to load "Dip8.mprt".

(b) Enter degree of rotation in "Enter Rotation Angle" dialogue. If you hit OK without entering a number, rotation angle is set as "0".

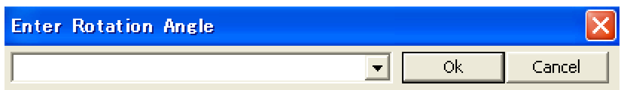


Fig. 14 Entering Rotation Angle

* Rotation angle is in counter clockwise direction.

(c) When the status bar says "designate first position", move the cursor to where No. 1 pin of the DIP should be located, then click. The display shows DIP 8 parts as in the screen shot. In the same manner, load "Sip2.mprt", and arrange it as shown in the screen shot.

* To undo the entry, hit Back Space.

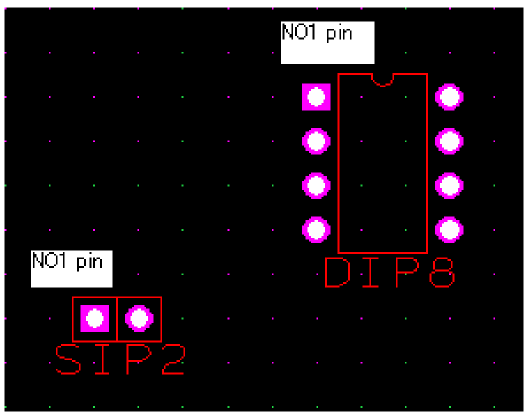
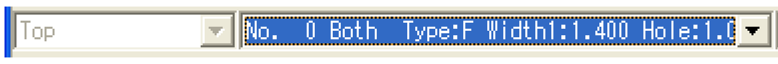


Fig. 15 Showing Dip8 and Sip2

(d) After arranging the necessary parts, hit ESC on the keyboard.

(2) Arranging pads

(a) Hit pad  icon in drawing tools. Select one of preloaded pads from "Size" menu



.

In this example, let's select No. 0.

(b)When the status bar says ""Specify the point for pad", move the cursor to where you want to have the pads, then click.

Arrange pads as in Figure 16.

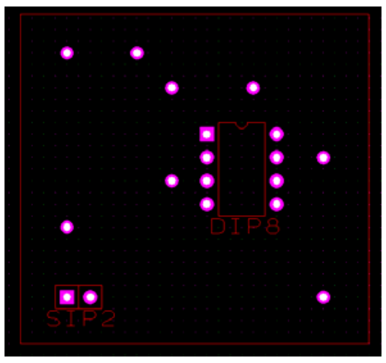


Fig. 16 Dip8 with added pads 1

＜Editing＞

You can edit the pattern using edit tools. Select an editing tool from the tools bar, and follow the instruction in the status bar.

Examples:

Transferring : Parallel Transfer, Rotational Transfer, and Flip Over tools are available.

Copying : Parallel copying, Rotational Copying, and Flip Over copying are available.

* Although there is no specific icon for "delete" function, you can group the items, then hit DEL from the keyboard to delete.

＜Example of Editing . . . Parallel Transfer＞

(a) Click parallel transfer icon  to show “Select an element” in the status bar.

(b) Group the elements to be transferred, or select an element to transfer by clicking it. Grouped elements are indicated with a white dotted line as shown in Fig. 17

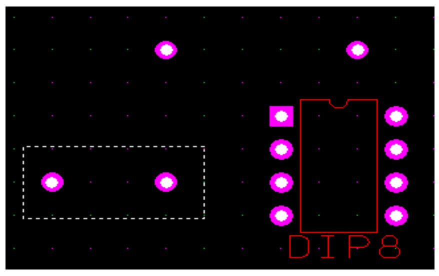


Fig. 17 Group Elements

(c) Right click to confirm the elements, then confirmation window pops up. Click “Confirm”.

Fig. 18 Confirming Group Elements

(d) The status bar displays “1. Select a Base Point”. Move the cursor to a pad to be used as a reference point, then click. The status bar now shows “2. Select a destination point”. Move the cursor to the point where the elements are to be moved, then click.

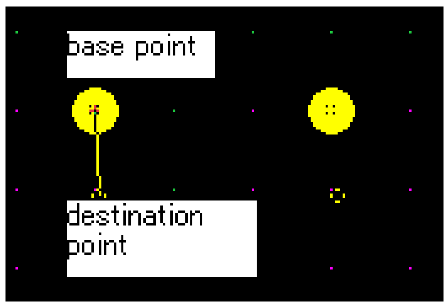
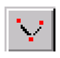


Fig. 19 Choosing a base point

<u>(3) Wiring Pattern</u>

Here, we will explain how to make wiring patterns on soldering surface of a single sided board.

① Click drawing tool icon  to make a line segment specifying two points.

② Choose the soldering surface by selecting 

from layer menu.
Select 

from the menu to choose pattern width and size.

③ The status bar displays “Specify a start point”, then move the cursor to the starting position of the line segment, then click. The status bar now changes to “Specify a end point”. Move the cursor to the end point, then click. A line segment is drawn.

* To end a line segment　: Right click or ESC from the keyboard.
* To continue line pattern :　Specify the next point.

④ To end a pattern work, hit ESC key on the keyboard twice. There should be no message in the status bar.

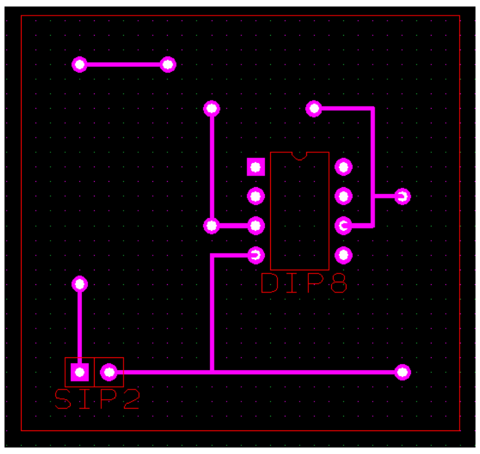


Fig. 20　Drawing line segments

＜Note＞

For smaller line spacing, such as drawing a pattern between IC pins, set grid spacing to 1.27 mm (= 2.54mm/2).

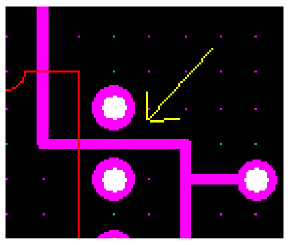
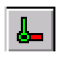


Fig. 21　Smaller line spacing

＜Note＞

You can also use line data tool  to draw wiring patterns.　Double click in a certain position of a line segment to change the pattern from top layer (parts surface) to bottom layer (copper surface). A via hole is automatically made.　To draw wiring patterns on bottom layer (copper surface), click at the starting position, then double click to move to bottom layer.

2.　Designing a new pads and patterns

Here, we will explain how to set up shape and size of pads and patterns to be used for drawing.

Pad : Components are soldered onto pads. Depending on the part is discreet or surface mount, the pad is drilled or not.

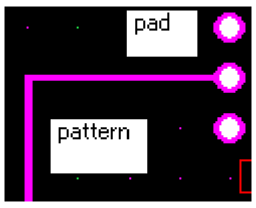


Fig. 22 Pads and Patterns

Pattern: Mainly used to connect a pad and another pad.

＜Process of making pads＞

Set up art work conditions. (for the area to be soldered)

＋

Set up drilling conditions

⇩

Set up pad conditions

＜Process of drawing patterns＞

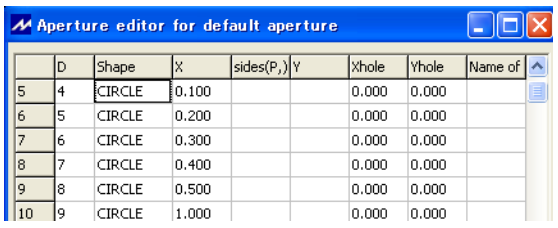
Set up art work conditions. (pattern width)

2-1. Drawing Pads

(1) Artwork Conditions

Select "Draw Prefs." / "artwork" to show "Aperture Editor" as shown in Fig. 23. Using this editor, you can set up shape, size, and pattern width.

① To change aperture shape, move the cursor to shape field, then, click mouse right button. Aperture information is based on RS274X standard format. In this example, set "D" as "4".


Aperture editor for default aperture

	D	Shape	X	sides(P,)	Y	Xhole	Yhole	Name of
5	4	CIRCLE	0.100			0.000	0.000	
6	5	CIRCLE	0.200			0.000	0.000	
7	6	CIRCLE	0.300			0.000	0.000	
8	7	CIRCLE	0.400			0.000	0.000	
9	8	CIRCLE	0.500			0.000	0.000	
10	9	CIRCLE	1.000			0.000	0.000	

Fig. 23 Aperture Editor

② To change aperture shape, select the first letter of each shape as shown in Fig. 24. In this example, select CIRCLE.

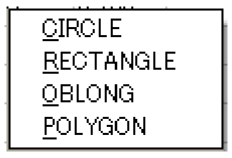
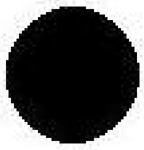
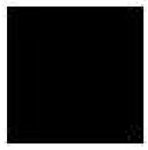


Fig. 24 Shape

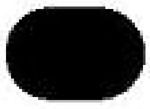
For selected shape, enter appropriate numbers for X, Y, width or number of sides.



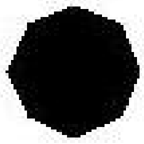
Shape：CIRCLE
X: Line width or Diameter of circular pad



Shape：RECTANGLE
X: Width
Y: Height



Shape: OBLONG
X: Width
Y: Height



Shape: POLYGON
X: Diameter
Sides: Number of sides

③ Move the cursor to "X", select it, and enter the number.
In this case, enter "1.6 (mm)". For rectangle or oblong, enter both "X" and "Y" values. When you have entered all the necessary numbers, hit DONE to end aperture editing process.

(2) Drilling Conditions

Go to "Draw Prefs." and select "Drill" to show Hole (T-Code) setting (Fig. 25). From this editor, you can set up drill size.

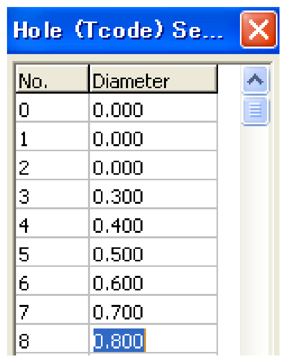


Fig. 25 Drill Setting

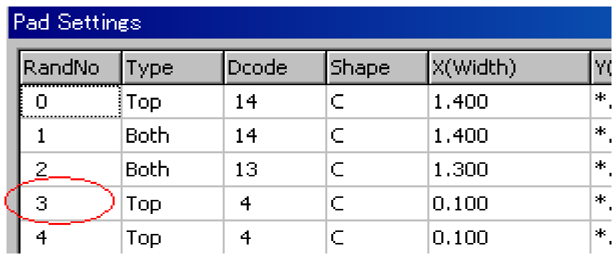
Typical drill sizes are already set as default. Unless you need a special drill size (for example, 0.55 mm) for your PCB, it is not necessary to enter drill settings.

① Move cursor to "Diameter" cell to change.
② Drill number is referred here as "No." In this example, highlight diameter for Tool No. "8".
③ Enter drill size. In this example, enter "0.8 (mm)". Click DONE to exit drill setting editor.

(3) Pad Conditions

Go to Draw Preference, then select "Pad" to show "Pad Settings" editor (Fig. 26). We will show how to make a "pad" using a shape described in (1) Artwork Conditions above, and a drill hole described in above (2) Drilling Conditions.

① Move cursor to Pad No to edit, then left click to select. In this case, select Pad No. 3.


Pad Settings

RandNo	Type	Dcode	Shape	X(Width)	Y(
0	Top	14	C	1.400	*.
1	Both	14	C	1.400	*.
2	Both	13	C	1.300	*.
3	Top	4	C	0.100	*.
4	Top	4	C	0.100	*.

Fig. 26 Pad Settings

② Use Tab key or mouse to select "Type", then right click. You can specify Top, Bottom or Both. In this case, choose "Btm" for bottom surface.

Top : Pad on top (parts) surface only.
Btm : Pad on bottom (soldering) surface only.
Both : Pad on both top (parts) surface and bottom (soldering) surface.

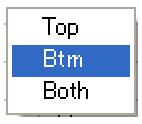


Fig. 27 Selecting PCB Surface

③ Hit TAB key to move the cursor to "D code", then left click. A pop up window shows available conditions set at (1) Artwork Conditions above. Select a shape of a soldering pad to be used. In this example, choose "No. 4 F Width 1.6".

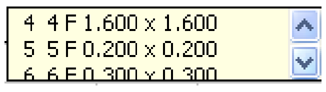


Fig. 28 Selecting Aperture

④ Hit TAB key to move the cursor to "T Code", then left click. A pop up window shows available conditions set at (2) Drilling Conditions above. Select the drill hole condition to be used. In this example, select "No. 8 Width 0.8". (Leave "T Code" setting as ***** if there is no drill hole.)
This completes the process of assigning "Pad No.3" as 0.8 mm drill hole with 1.6 mm diameter soldering pad as shown in Fig. 30.

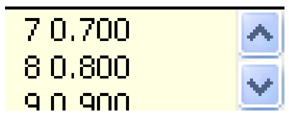


Fig. 29 Selecting Drill Condition

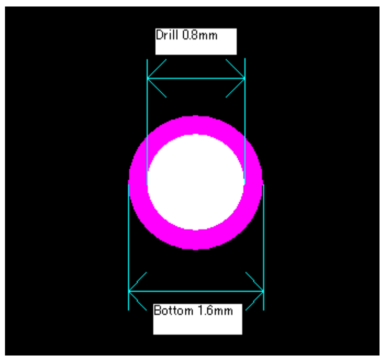


Fig.30 1.6 mm Pad

- You can save the conditions set through procedures (1) ~ (3) above from "Draw Prefs." / "Save Conditions" as a template (*.CND) file.

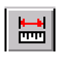
II． Generating PCB milling data

Once the pattern layout is done, we need to generate milling data or contour line.
We will also prepare outline data.

1． Contour Extraction

(1) Considerations on Contour Width

An appropriate contour width should be set with pattern and pad clearances into consideration. You should measure the nearest distance in your pattern. In this example, pin to pin distance of DIP 8 is the shortest distance in the circuit pattern.

① Click “Display Distance” icon  on the menu bar.

② The status bar says “Select a first element”. Click No. 1 pin of the DIP8 component. In a popup menu shown in Fig. 31, select “flash/drill”.

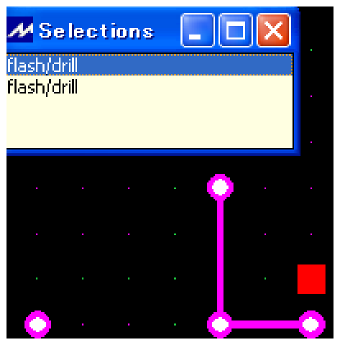


Fig. 31 Selecting an element

③ The status bar changes to “Select a second element”. Click No. 2 pin of the DIP8 component. In a popup menu shown in Fig. 32, select “flash/drill”.

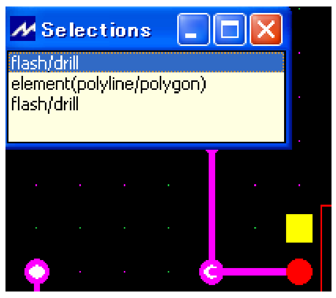


Fig. 32 Selecting second element

④ Distance popup window (Fig. 33) shows:

Distance: Distance between the center points of element 1 and element 2.

Nearest: Distance between the nearest points of element 1 and element 2.

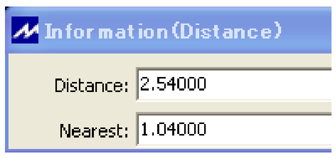


Fig. 33 Distance Popup

※ The width of contour milling should be set at less than “nearest distance” measured above. In this example, it should be less than 1.04 mm.

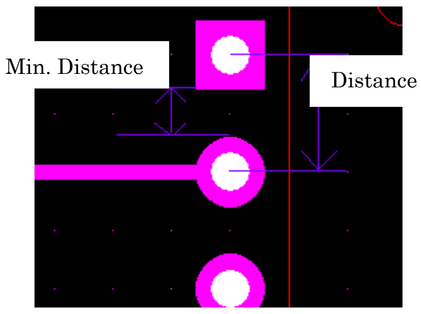


Fig. 34 Minimum Distance

Caution: If you choose a milling width larger than "minimum distance', some pads may be eroded.

(2) Contour Extraction

① Click "Contour Extraction" tool to show "Generation Settings" sub menu (Fig. 35) and confirm/enter the followings.

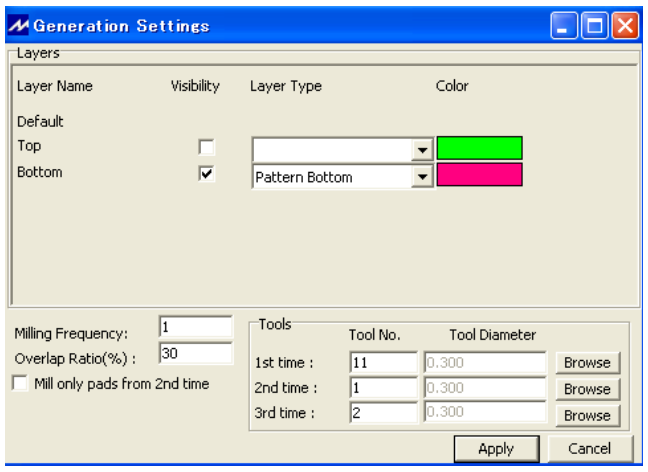


Fig. 35 Generation Settings

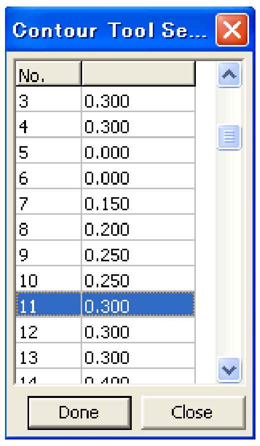


Fig. 36 Contour Tool Setting

② Layer Name : Confirm if Bottom is checked.
③ Layer Type : Confirm if Pattern Bottom is selected.
④ Milling Frequency : Confirm if it is set as 1.
⑤ Tool No. : Under "Tools" category, hit Brouse for "1st time" to show "Contour Tool Selection" popup submenu.
Select an appropriate tool to meet the following conditions.

* The width should be less than the nearest distance measured in (1) above.
* Tool number should be No. 5 or higher.
(Tool Numbers 0 to 4 are used for other process.)
* Tools to be used.
(Basically, milling cutter has a 90 degree tip and used for milling widths of 0.2 mm to 0.4 mm.)
Considering above, we select "No 11 0.3 mm" for this time. Hit Done to select the tool.

⑥ When you hit Apply button, a contour line (dark pink) is extracted around patterns(pink). The PCB prototyping machine cuts along this contour line (dark pink) with a selected tool.

* To change the color for each pattern or contour, select the layer in "layer" menu, right click, and "property" to show "Define each layer" submenu. By clicking colored square, you can define your color of choice.

＜Setting up contour extraction parameters (refer to (2)① above)＞

If you are concerned that 0.3 mm width might cause soldering bridge, you can make cutting width wider by cutting contour repeatedly. To process contour twice, enter "2" in "Milling Frequency" box. In "Tools" box, select the same tool (No11 0.3 mm) for "2nd time" tool. Enter 30 (%) for "Overlap Ratio (%)". In this setting, cutting width becomes:

0.3 mm (cutting width) x 2 (times) – (0.3 mm x 30 % (Overlap Ratio) x (2-1)) = 0.51 mm

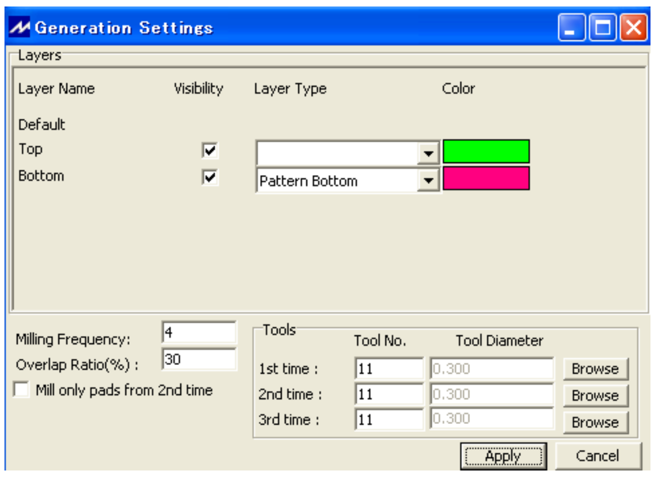
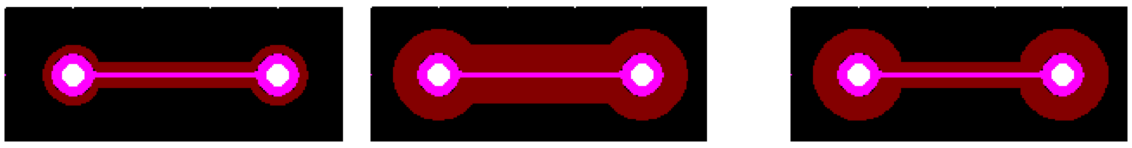


Fig. 37 Setting Contour Extraction Parameters

To repeat milling three times, enter "3" in "Milling Frequency", then, select Tool Number for 3rd time. If you repeat milling more than 3 times, the tool number for 3rd time shall be used for the 4th and later cutting as well.

Check "Mill only pads from 2nd time" to make redundant cutting only around pads.



Milling Frequency: 1　　Milling Frequency: 4　　Milling Frequency: 4 (Only around pads)

Fig. 38 Overlapping Milling

With overlapping milling, soldering becomes easier. On the other hand, it takes more time for milling work, and tools wear out more. Use appropriate overlapping depending on the patterns to work on.

＜Changing/Redoing contour extraction＞

(a)To change contour extraction setting follow the procedures in (2) ① above to bring "Contour Extraction Setting" menu shown in Fig. 37. Enter the parameter to change, then click Apply to show "Confirmation" screen shone in Fig. 39.

Fig. 39 Confirmation

Click an appropriate radio button in the confirmation screen, then hit Apply to delete the previously generated contour lines and redo contour extraction based on the new parameters.

All	:	Outline and Rubout
Outline	:	Outline
Rubout	:	Rubout

２． Generating outline (routing) data

To generate outline (routing) data, use board outline prepared in Section Ⅰ 1-2(1) above. From Layers pane, select "Default", then right click and show Properties popup menu. Choose "Add" as shown in Fig. 40.

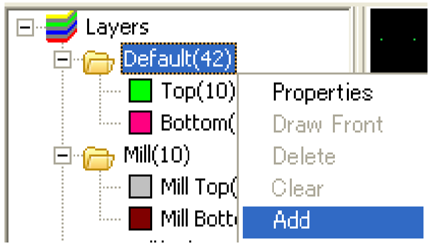


Fig. 40 Adding a layer

Enter "Boardoutline" in "Specify a layer Name" screen, then hit OK. (Fig. 41)

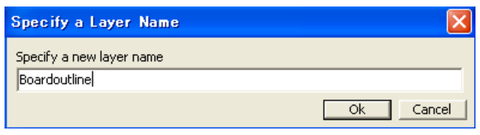


Fig. 41 Entering a layer name

A new layer named "Boardoutline" is now shown in "Layers" pane. (Fig. 42)

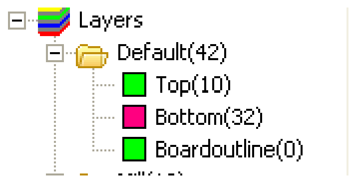


Fig. 42 New layer

In the working area, double click on a segment of the red outline. Choose "line (a segment) in the "Selections" popup menu. (Fig. 43) A red line segment changes to yellow line segment. In the same manner, change the other three line segments to yellow. Drag yellow rectangle to "Boardoutline" layer in the "Layers" pane.

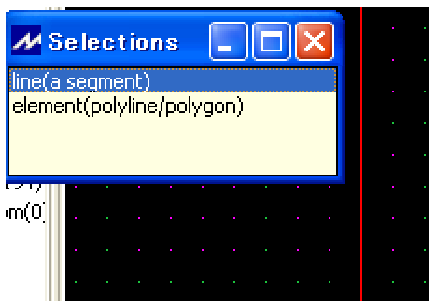


Fig. 43 Selecting line segment

You can see that the red line segments have changed to green. Note that number of elements in "Boardoutline" layer has also changed from (0) to (4).

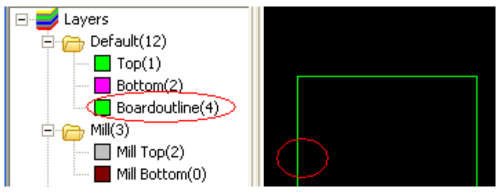


Fig. 44 A line segment moved to "Boardoutline" layer

Now, click Routing tool shown in Fig. 45.

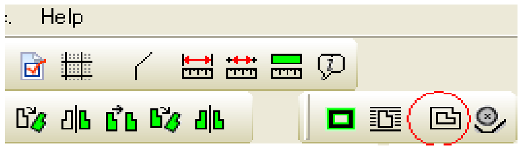


Fig. 45 Routing Tool

Clicking routing tool brings up "Generation Settings window shown in Fig. 46. You can enter routing parameters from this window.

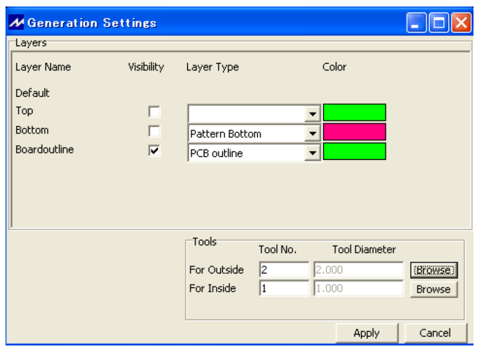


Fig. 46 Setting up routing parameters

Click "visibility" box for "Boardoutline" layer, and select "PCB outline" as Layer Type.
In Tools box, click $\boxed{\text{Brouse}}$ button for "For Outside" to bring "Outline Tool Setting" submenu (Fig. 47).

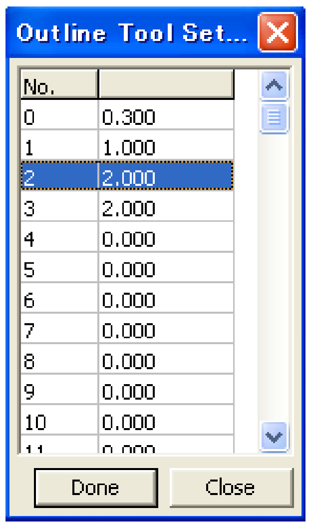


Fig. 47 Outline Tool Setting

Use a router (forming cutter) for cutting PCB outline. Consider tool size when you are generating outline data.

In this example, choose tool number 2 with tool size of 2.000 as shown in Fig. 47.
Click $\boxed{\text{Done}}$, then click $\boxed{\text{Apply}}$.

An outline data is generated as thick aqua line shown in Fig. 48

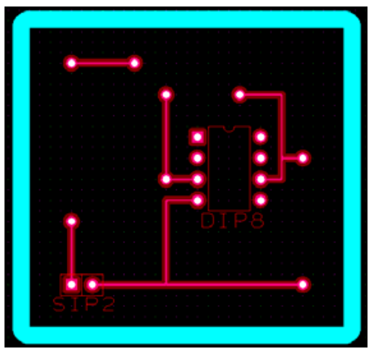


Fig. 48 Routing line

3. Generating hatching data

Hatching function is used to remove copper layer around circuit pattern. When you need to hatch around circuit pattern, you should extract milling data first, then, generate hatching pattern around it. To start hatching data generation, click hatching icon on the menu as shown in Fig. 49.

Fig. 49 Hatching tool

When you hit hatching icon, "Rubout Generation Settings" sub screen pops up (Fig. 50).

In this example, set parameters as:

① Rubout for : Check Bottom

② Rubout Tools:

(We use small and Medium size tools in this example.)

Small size: No. "0", and 0.300

Medium size: No. "20" and 1.000

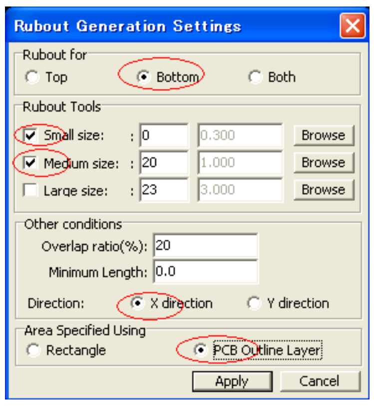


Fig. 50 Rubout Generation Seetting

③ Other conditions:
Overlap ratio (%): 20
Direction: X
* For hatching, overlapping of 20% is recommended.
Direction is typically set for “X direction”, but depending on circuit pattern or machine type, “Y direction” is also used.

④ In “Area Specified Using” box, check “PCB Outline Layer”.

⑤ After setting above, click Apply to start hatching (rubout) data generation. Fig. 51 shows hatched pattern.

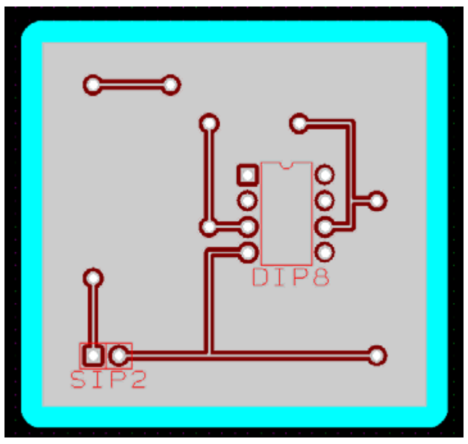


Fig. 51 Hatching data

4. Saving a data file

To save the PCB milling data, follow the procedure shown below. The suffix of data file is mit (***.mit). Go to “File” pull down menu to use “save as” function. Select appropriate folder, enter a file name, and click SAVE button. The “File Information” screen pops up. Enter file information or comments if necessary, then click OK to save the data.

5. Printing

To print out PCB milling pattern, follow the procedures shown below (Fig.52).

(1) Go to menu to select [File], then [Print]. Click Print layers button to show “Layer Settings” screen.

(2) Select layer to print, then, click OK button.

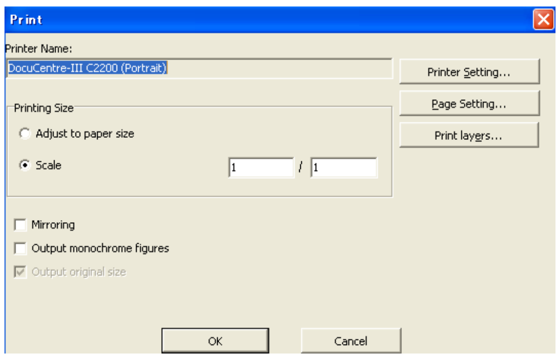


Fig. 52 Printing PCB milling pattern

(3) Use “Printer Setting” to select a printer and set up printing parameters. Use “Page Setting” for page setting. Click OK to print.