MITS Design Pro

CONVERTER EASYCAD

Revision

091208 Bug Fixed: require Acrobat Japanese font pack

Contents

- Polygon Fill
- Multiple Polygon Fill

Importing Examples

- Altium Designer (Protel) RS274X
- Cadence OrCAD PCB Editor RS274X
- Cadence OrCAD PCB Layout RS274X
- Cadsoft EAGLE RS274X
- CSi WinPCB RS274X
- Designsoft TINA RS274X
- Mentor Graphics PADS RS274X

Drawing with EASYCAD

- Grid settings
- Draw Boardoutline
- Pad
- Line&Pad
- Print
- Polygon Fill
- Multiple Polygon Fill
- SavePart
- PlacePart
- Change Attribute
- Move Layer from Top to Bottom
- Enlarge
- Fonts
- Text
- Text to milling

Generating Milling data

- Measure Distance
- Auto Drill
- Generate Outline
- Generate Outline (Negative)
- Generate Rubout
- Generate Routing
- Milling Through Gap

Basic Information

Install Software

MITS Design Pro is a software for PCB design, converting data from other CAD system and controls the machine.

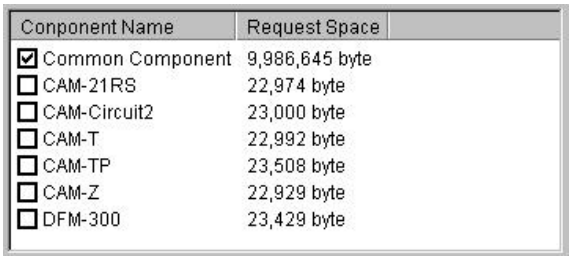
PC Specfications:

OS: Windows 2000(SP4 and newer edition)/XP Home(SP2)/XP Pro(SP2)/ Vista
NOT supported in 64bit operating system.
RS232C or USB port is required.

Install Procedures:

Open CD-ROM and then start Designpro_en_setup.exe to install the program.

Choose the appropriate items in the list during the set-up procedure:

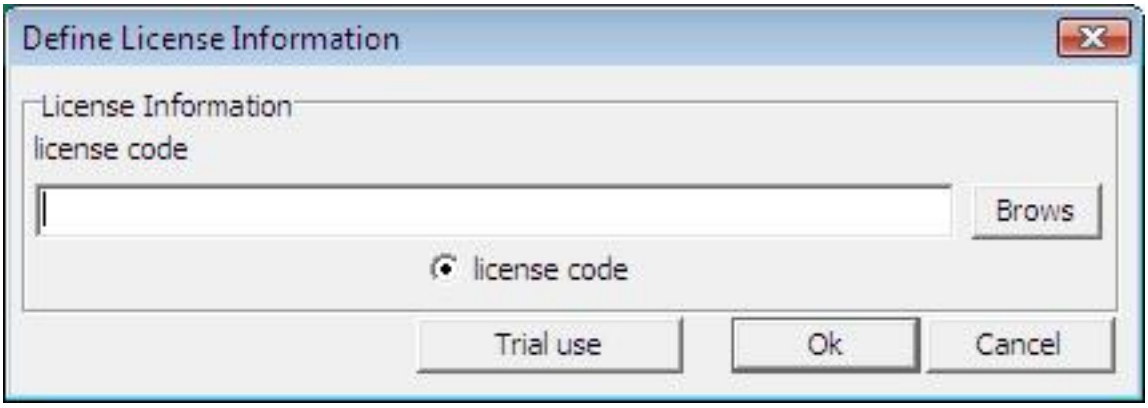


*Common component must be installed together with the applications. It includes Converter and EASYCAD.

Choose one of the following applications related to your board maker:
*CAM-21RS controls FP-7A / FP-21A < including HP option>.
*CAM-Circuit2 controls Seven mini / Eleven-T < including HP option>.
*CAM-TP controls FP-21T Precision.
*CAM-T controls FP-21T < including HP option>.
*CAM-Z controls Eleven Auto / FPZ-31AT / FPZ-73AT.

Registration

Registration is necessary to continue to use MITS Design Pro and board maker. The following screen is displayed when MITS Design Pro is started.



Please enter the license code and then click OK to finish the registration.

Otherwise, click "Trial use" to evaluate the software for 1 month.

MITS Electronics submit 2 type of license code:
- Activates optional EASYCAD
- No optional EASYCAD (Converter+CAM ONLY)

License Policy:
One boardmaker includes one license of Mits Design Pro software.
With this one license, you can install it to a PC for the control of the machine, and to another PC for making milling data for the machine.
When you install to 2nd PC, enter same license code of 1st PC.

If you need more license, please purchase one license on one PC.

Start Up the Software

To start up MITS Design Pro:

[Start] -> [Program] -> [Mits Design Pro] -> [Design Pro]

To start up Correct DXF (Optional):

[Start] -> [Program] -> [Mits Software] -> [Correct DXF]

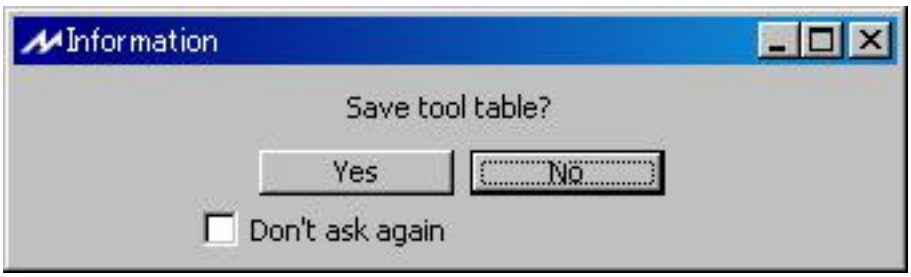
Exit the Software

To exit the software:

click on Exit on File menu, or click on the Close Box at the right corner of the title bar.

Save tool table?

If you prefer to save tool tables to the next time,
click Yes.



The following settings are saved :

- Settings in Tool tables menu
- Format options in Conv Prefs.menu

If you prefer to load the standard tool tables each time you start the software, click No.

If you check [Don't ask again] and then choose Yes or No,
the dialog don't appear from the next time.
When you want to change the settings, please go to Work Prefs. -> System.

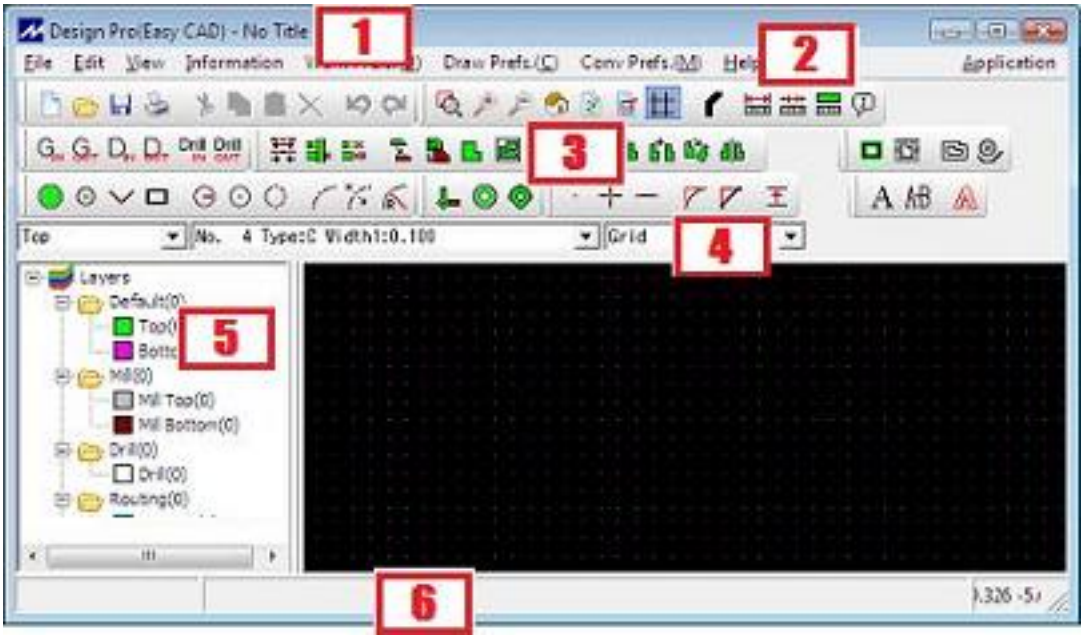
Changing Applications

To switch to another application, choose the desired application in the Application menu at the upper right corner of the screen.
MITS Design Pro applications are as follows:

EasyCAD : Used for designing PC boards.
Converter : Converts Gerber, DXF.
CAM-*** : Mills the board.

Screen

MITS Design Pro and CorrectDXF screen is as follows:



1. **Title bar**
   Displays the File Name during the work on the current file.

2. **Menu bar**
   Displays the application menu.

3. **Tool bar**
   Application functions are indicated by icon.
   Functions are activated by clicking on them.
   Tip help appears when the mouse corsor approaches the icon.

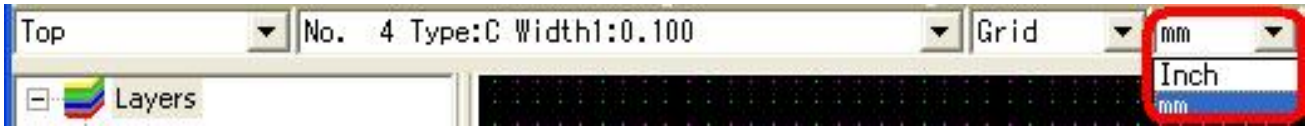
4. **Mode Settings bar**
   *lets you set the layer and line No. you wish to use.
   *lets you change the mouse recognition mode (Free, Grid, Point, Auto, On Line) .
   *lets you change the unit of values (mm, inch)

5. **Layer Panel**
   *lets you change the display of layers.(visibility, color and so on)
   *lets you browse and change the tables of apertures and tools.
   *lets you add and delete the layers.

6. **Status bar**
   Instructions and messages concerning the operation appears in this space.

Change Unit (mm/inch)

Unit can be changed in the mode settings bar.



When Gerber file or DXF file is imported, the unit mode is automatically changed according to the unit described in the Gerber/DXF file.
Please change unit manually if necessary.

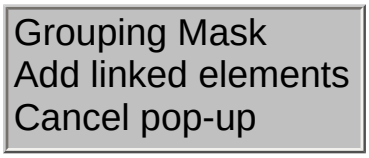
Mouse Operation

What is happened by clicking left or right of mouse buttons depends on the situations you are working.

When specifying a location:
Normally, the left button is used to specify a location on the screen when executing a command.
Use the right button to cancel a specified location.

Selection:
Click on an element or drag rectangle so that element(s) change its color. It is "selected".
Press ESC key to cancel the selection.

Right click to display pop-up which allows you the advanced selection methods:



When cutting a line:
Use the right button to cut a line when drawing a straight line or other type of line.

Entering Coordinates

When specifying points to indicate the locations of elements, home points and the like when creating or otherwise working with drawings, you can specify the point by clicking on the screen, or you can enter the coodinates of the point through the keyboard as follows:

Press the TAB key and the Enter Coordinates screen appears.

There are three possible ways to specify points through the keyboard. The distance in units can be expressed in millimeters or inches. You can select the desired unit using the Mode Settings bar.

1. Enter the *x,y* coordinates from a Relative point. For example, enter:

   100,100
   The point specified will be the one that is 100 units along the *x* axis and 100 units along the *y* axis from the point that was entered in the immediately previous operation.

2. Enter the *x,y* coordinates from an Absolute point. For example, enter:

   @100,100
   The point specified will be the one that i 100 units along the *x* axis and 100 units along the *y* axis form point 0,0 (the absolute point).

3. Enter Relative Distance plus Arrow key to indicate direction. Example:

   100->(Enter the distance in units and then the desired direction with the arrow key.)
   A + mark will appear at the point 100 units from the relative point (the point that was entered in the immediately previous operation) in the direction indicated by the arrow. If this point indicated by + is acceptable. confirm by clicking OK or pressing the Return key.

Layers

In MITS Design Pro, there are 2 kinds of layers for pattern and Boardmaker.

Layers for Pattern:
These layers are prepared for drawing PCB layout. When the program is started, there is a couple of Top and Bottom as default.
The layers are added each time Gerber and DXF data is imported from other CAD system. The different patterns with different apertures are displayed at the same time in a screen.

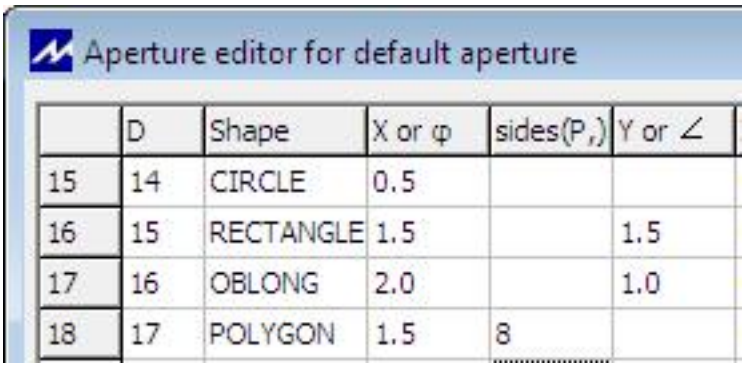
Layers for Boardmaker
Milling layer, Drill layer and Routing layers are layers for boardmaker.
These layers cannot be added because of the operations in CAM application.

Relationship of Folder and Layer:
Each layer exists under its own folder and each folder has aperture table or tool table as properties.:
The folder in which has layers for pattern has its aperture table.
The folder in which has layers for boardmaker has milling tool table, drill table or routing tool table.
Right click upon the folder and then click Properties displays table to be browsed and edited.

Aperture and Tool Settings

Basically, aperture tables and tool tables can be called up from the layer panel. Right click -> Properties upon the folder will display the tables.

Aperture Table (Artwork)



	D	Shape	X or φ	sides(P,)	Y or ∠
15	14	CIRCLE	0.5		
16	15	RECTANGLE	1.5		1.5
17	16	OBLONG	2.0		1.0
18	17	POLYGON	1.5	8	

Right click upon Shape field will display the pop-up to change the aperture shape. Aperture shapes are based on the standard apertures of RS-274X format.

CIRCLE
RECTANGLE
OBLONG
POLYGON

Enter X, Y or Sides according to the aperture shape.

Shape: CIRCLE

X:Line width or Land diameter

Shape: OBLONG (OVAL)

X: X dimension **Y:** Y dimension

Shape: RECTANGLE

X: X dimension **Y:** Y dimension

Shape: POLYGON

X: Outside dimension **sides:** number of sides

Tool Table (Milling, Drill, Routing Tool) :

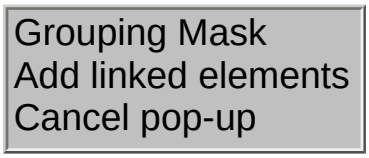
Tool table has no field of "Shape". Enter the diameter only.

Select Elements

The following operations let you to select the element(s).

Selection:
Click on an element or drag rectangle so that element(s) change its color. It is "selected".
Press ESC key to cancel the selection.

Right click to display pop-up which allows you the advanced selection methods:



Select Elements and then Edit
When the element(s) are selected, you can use the following commands to edit them or show their properties.

Delete, Copy (Parallel, Rotate, Mirror), Move(Parallel, Rotate, Mirror),
Change Properties, Show Properties

Appendix:
If these commands are clicked with nothing selected, it is also available. The commands will prompt you to select element(s) and then right click to comfirm the selection will let you proceed the next step.

Drag & Drop the selected element(s)
If the selected element(s) are dragged and then dropped in the drawing area, the location of the element(s) will moved.
If the element(s) are dropped in the layer panel, the element(s) will be moved to the layer.

Hide Layer:
Elements in hide layer cannot be selected.
This rule makes it easier to set a layer hide before editing when there are elements which is not necessary to be edited.

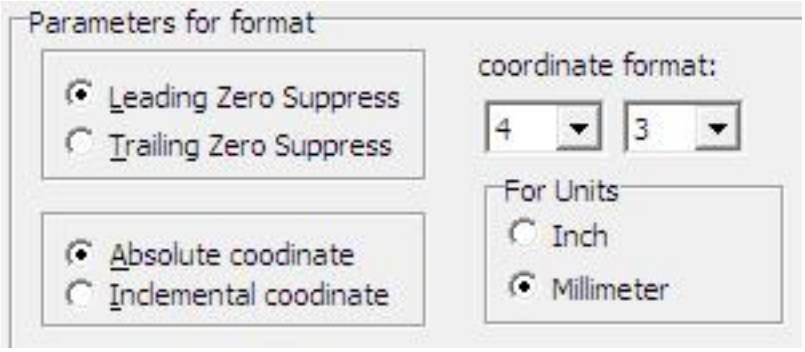
Importing Gerber File

Gerber IN

Drag and Drop Gerber files into MITS Design Pro screen and import process is started.

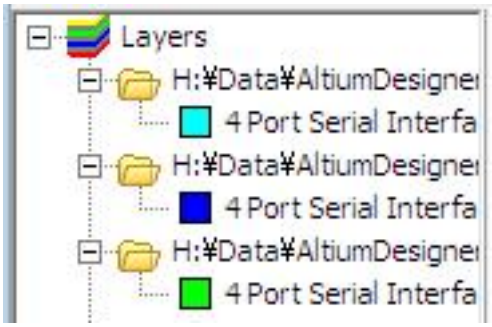
Otherwise, Choose File -> Import -> Gerber IN
Select the file and click Open.

Gerber Setting:
This screen let you comfirm and change the settings of Gerber format and apertures.



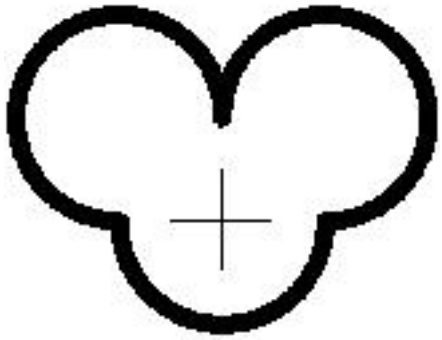
When the Gerber is RS274X format, the settings are displayed in this screen as written in the header of Gerber file.
It is not necessary to change them.

When the Gerber is RS274D format, the default settings are displayed in this screen.
Change them to the correct settings, otherwise the data will be loaded incorrectly.



Each time the gerber file is imported, the new layer will be added.

Single-Quadrant mode:
Usually check off this option.
When you import the old gerber file from Orcad and arcs are converted into the wrong trace, this option will help to convert correctly.



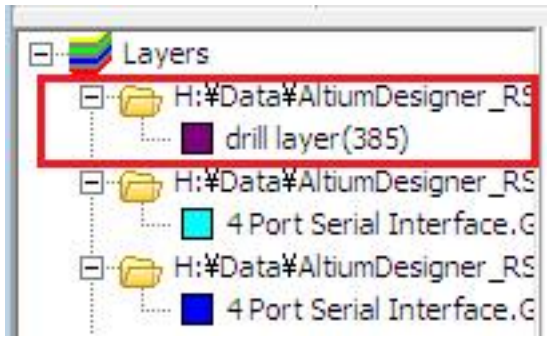
Drill IN

Drag and Drop NC Drill file into MITS Design Pro screen and import process is started.

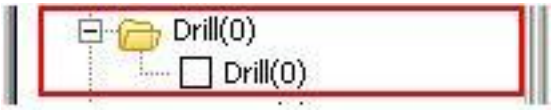
Otherwise, Choose File -> Import -> Drill IN
Select the file and click Open.

New (Recommended):
When New is selected, a layer will be created and drill data will be imported into the layer.

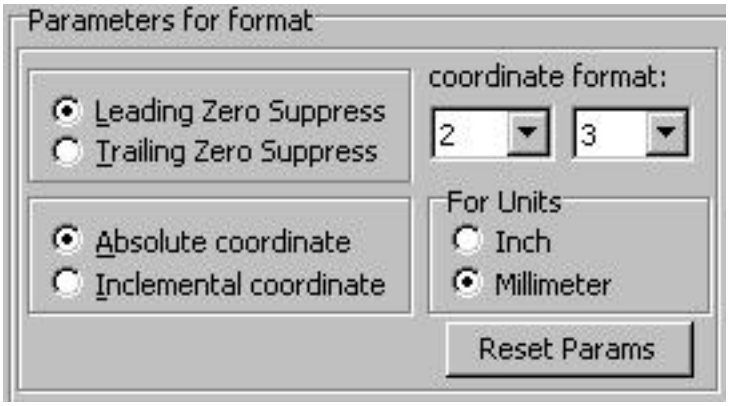
After that, Auto drill function will re-assign the tool No. of holes to the standard tool table of the machine.



Direct to drill layer:
When this option is selected, drill data will be imported into the existing drill layer.
That's the same way of the previous version Flashwin.



NC Drill Setting:
This screen lets you comfirm and change the settings
of NC Drill format and tool diameter.



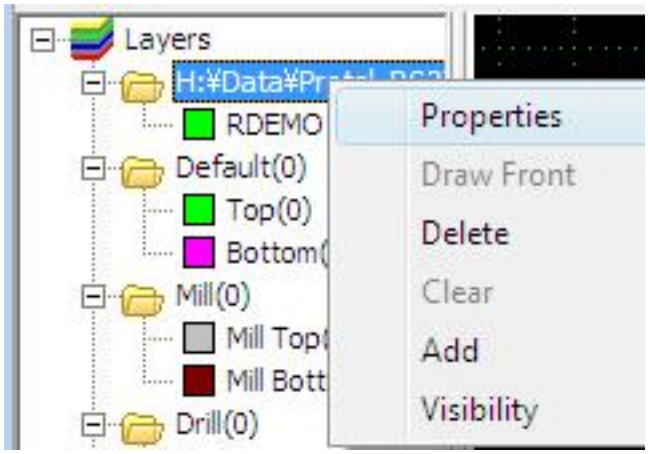
Autodetecting button detect the most suitable parameters of drill format according to the Gerber pattern which is imported previously. Autodetecting button is one of the features of this software, however, this button doesn't appear when no Gerber pattern is found for reference.

Load Aperture List

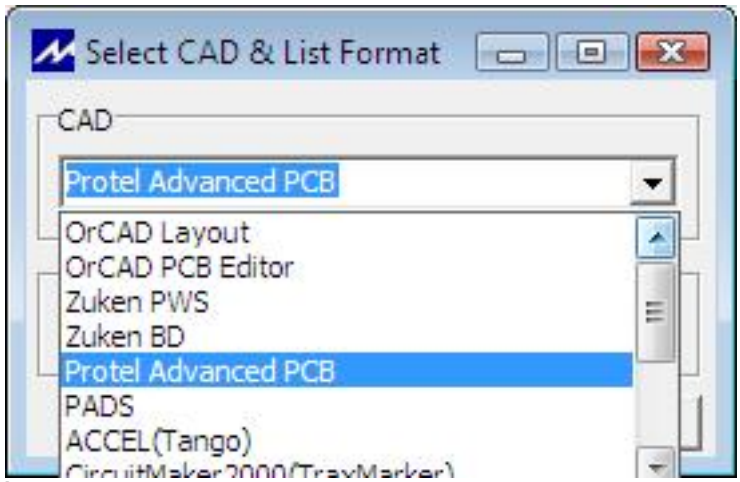
When gerber data is RS274D format, you need to input aperture manually. However, this program provides the load aperture function to help input works. Load Aperture supports some of aperutre list and drill list from CAD system.

The operation of loading the aperture list is as follows:

Basically, aperture tables can be called up from the layer panel.
Right click -> Properties upon the folder will display the table.



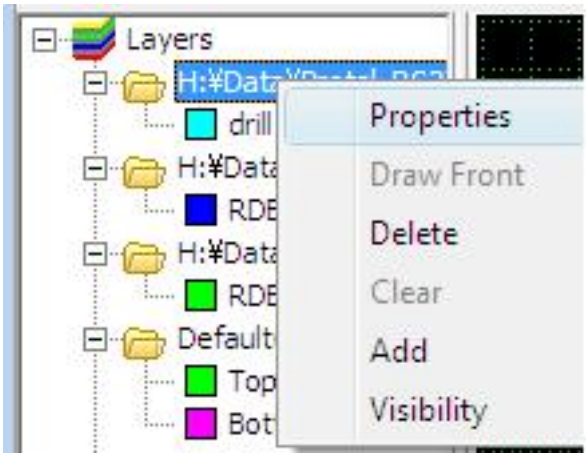
In Aperture Editor screen, click Aperture file.
Load Aperture screen will be displayed.



Choose CAD name in the CAD List, click on Aperture in List Type, and then click OK.
In the next screen, choose aperture list and then click OK.
The program will read the aperture list and then refrect it to the aperture screen.

The operation of loading the drill list is as follows:

When NC drill data is imported into the new layer, a drill layer is added on the layer panel.
Right click -> Properties upon the folder will display the table.

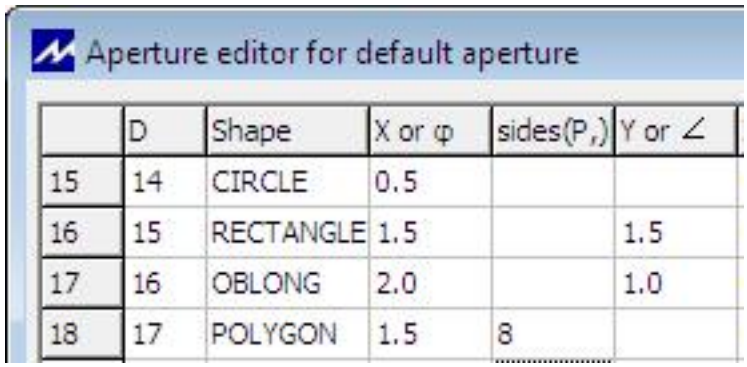


The rest of operations is same as loading aperture list but
to click on Drill in the List Type field.

Conversion Trouble Shooting - Gerber / NC Drill

The shape and width of lines and pads are different.

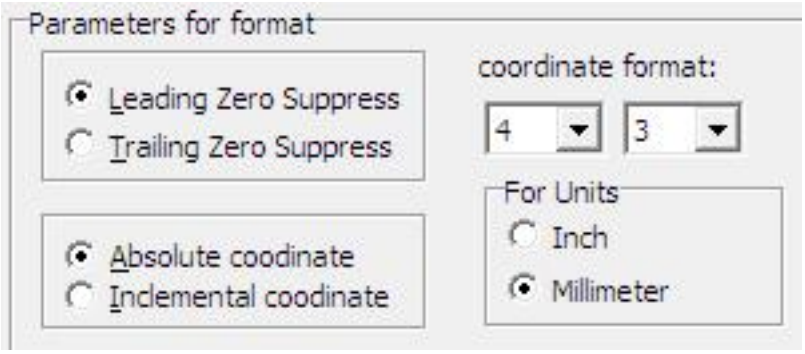
Check the settings of the aperture and drill table.



	D	Shape	X or φ	sides(P,)	Y or ∠
15	14	CIRCLE	0.5		
16	15	RECTANGLE	1.5		1.5
17	16	OBLONG	2.0		1.0
18	17	POLYGON	1.5	8	

The size of the drawing is different.

Check Cordinate Format (integer and decimal value) and Unit (inch or mm).



Lines appear that completely different from the pattern drawing.

Check either Absolute or Incremental.

Some parts are normal, but some lines and holes seem completely wrong.

Check Zero suppress.

Regarding Gerber data, mostly Leading Zero Suppress is used.

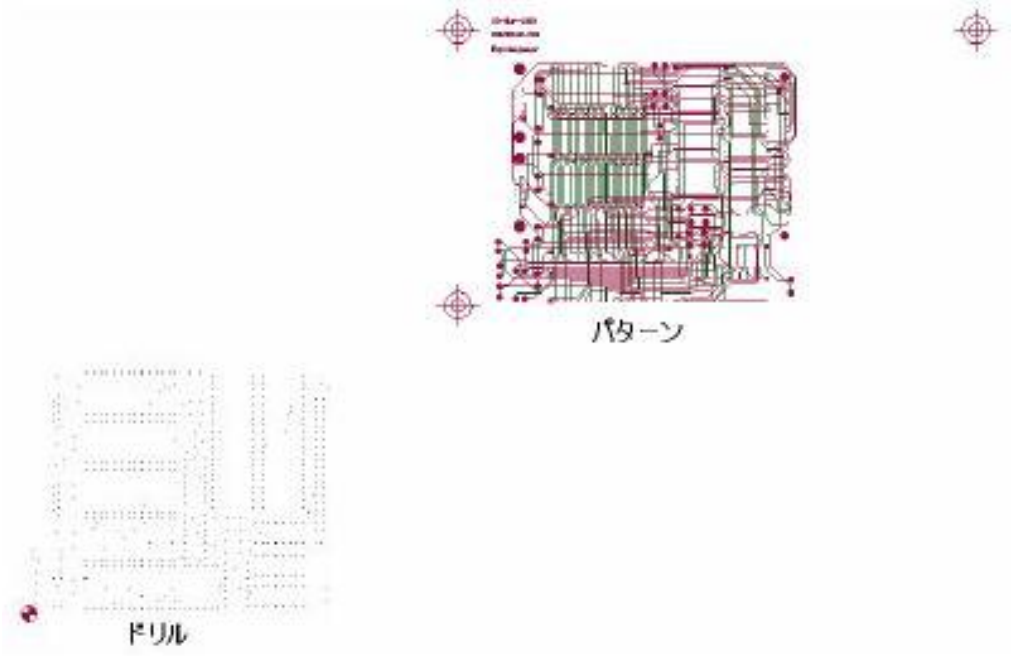
Regarding NC drill data, Trailing Zero Suppress is used in some CAD system.

When Trailing Zero is suppressed, it means that any last zero(s) in a value are dropped when the value is displayed.

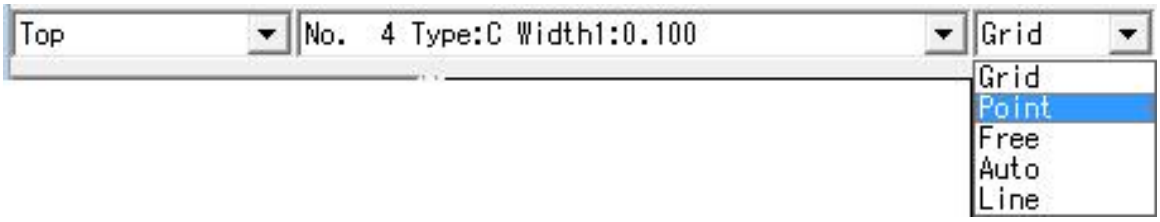
With trailing zero(s) suppressed, they are displayed as...	X00127Y00254 X001Y0015 X0035Y00042 X0825Y0026
With trailing zero(s) *not* suppressed, they are displayed as...	X00127Y00254 X00100Y00150 X00350Y00042 X08250Y00260

Move Holes

In the event that the locations of holes and patterns do not coincide, you can move the locations of holes so that they align correctly.

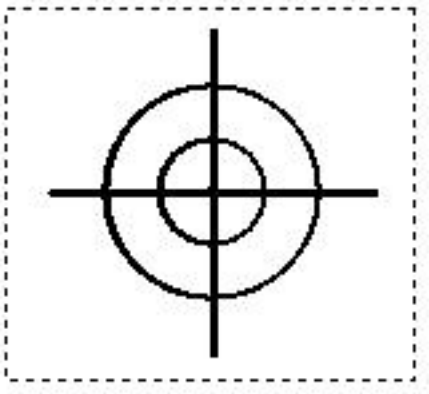


1. Drag rectangle with mouse so that holes are selected.
2. Set "Point" in the mode settings bar.



3. Click this icon. (Move-Parallel)
4. Click on any one hole to become the reference for moving holes.
5. Click on the location where the holes are to be moved to, all the hole data will be moved accordingly.

Delete

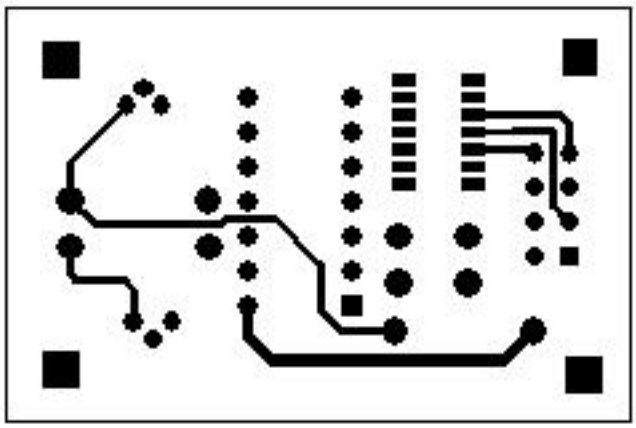


Click on an element or drag rectangle so that element(s) change its color. It is "selected".

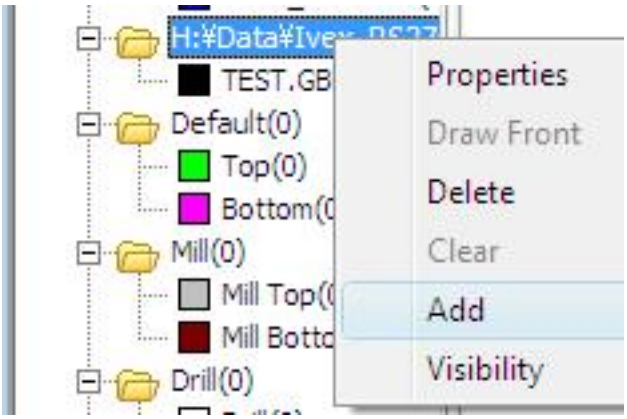
Click this icon to delete the selected data.

Move Board Outline to Another Layer

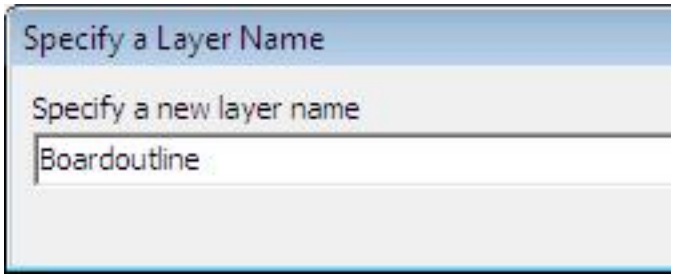
Move board outline to another layer when outline is drawn in the same layer of pattern.



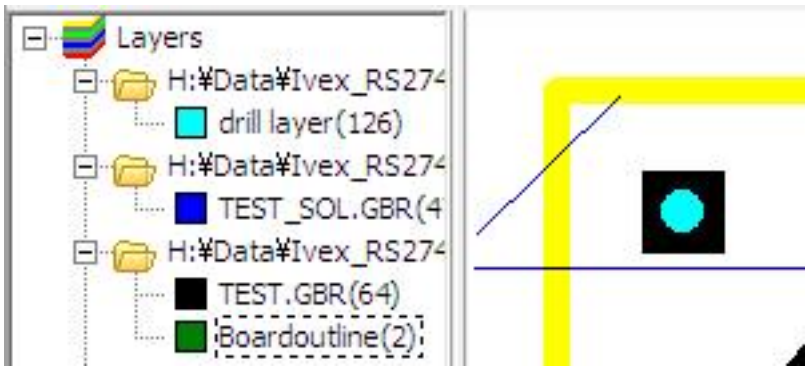
Right click on the folder in the layer panel and then choose Add.



Enter layer name and then click OK to add another layer.



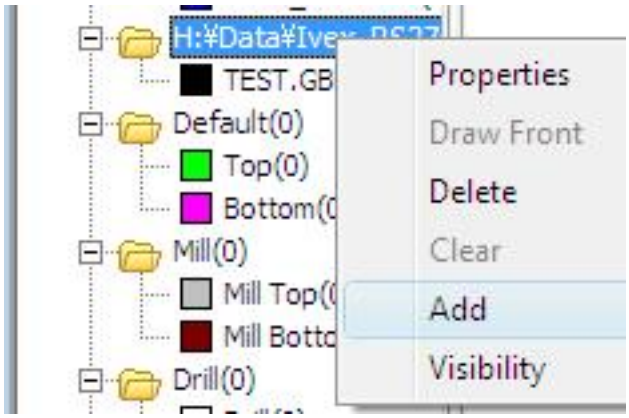
Select the outline and then drag'n drop it into the layer added in the layer panel.



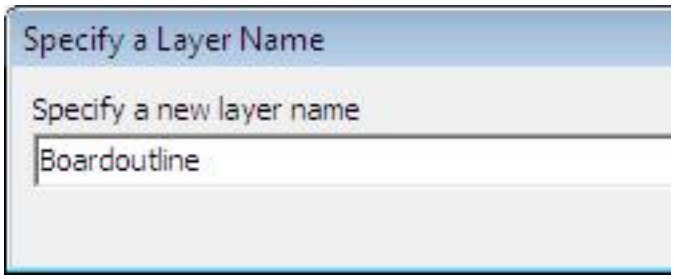
Draw Board Outline

When no board outline in the drawing from Gerber and DXF, or when you start the new drawing with EASYCAD,
add the layer to draw the board outline.

Right click on the folder in the layer panel and then choose Add.



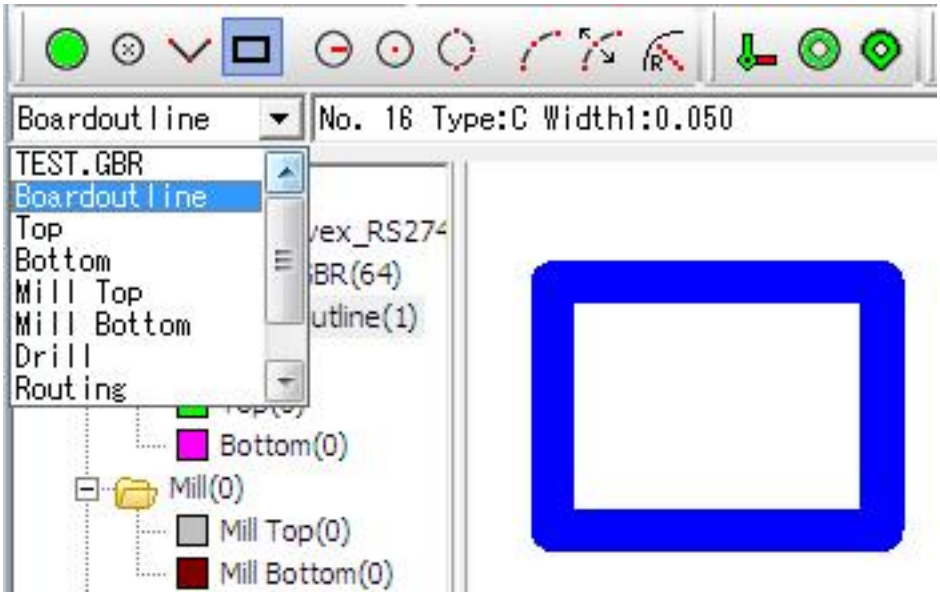
Enter layer name and then click OK to add another layer.



Choose the layer in the mode settings bar.

This icon lets you draw the rectangle.

Draw the rectangle by clicking on two diagonally opposite points.



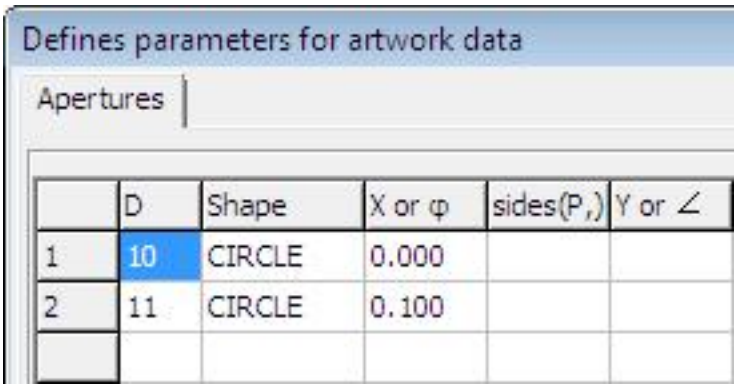
Importing DXF File

DXF IN

Drag and Drop DXF file into MITS Design Pro screen and import process is started.

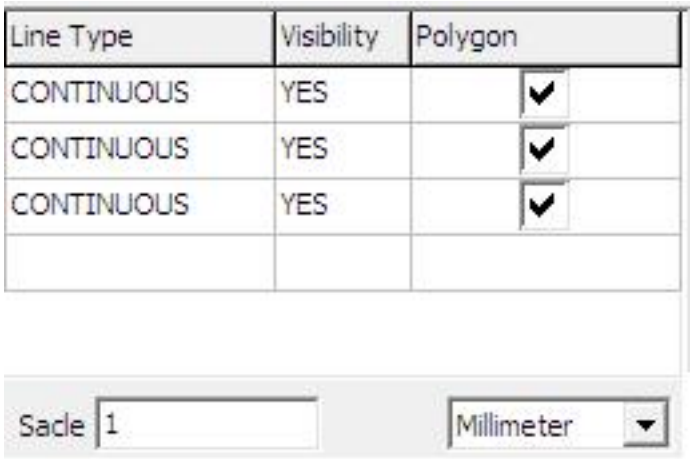
Otherwise, Choose File -> Import -> DXF IN
Select the file and click Open.

Aperture Setting:
Basically, DXF data has no width of line. However, data need the line width like Gerber data in order to be milled.
This Converter program put D-code to DXF data in the following rules:



DXF data	D code
LINE,ARC	D10
POINT	D11

Polygon and Scale:
This screen lets you to select 2 options:
- Polyline and Circle is to be converted to filled polygon.
- Scale

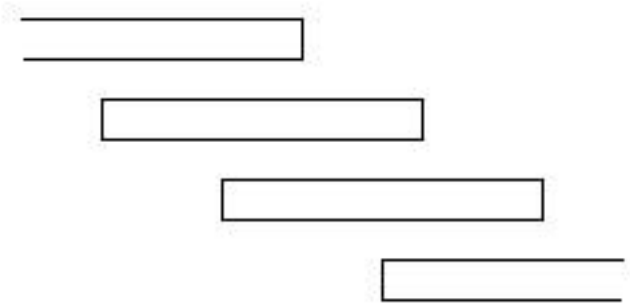


If Polygon field is checked, Polyline and Circle in DXF data is converted to filled polygon.
Filled polygon means "copper".
If Polygon field is not checked, these area is not filled inside and segments are converted to D10 lines.

Also, the data size is converted according to Scale and Unit field.

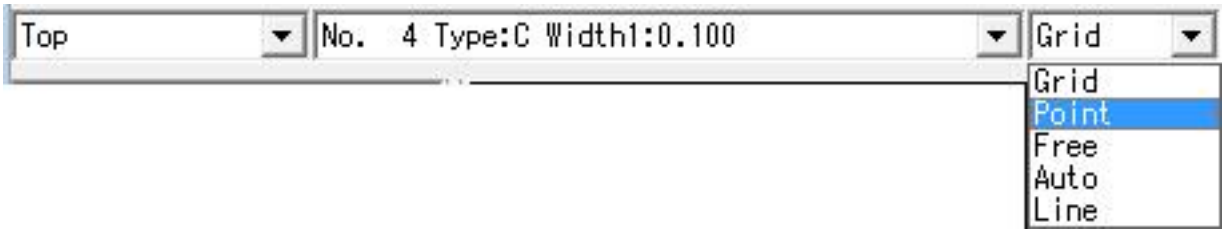
Draw Line

Let's connect the inturrupted ends with line in the illustration below.

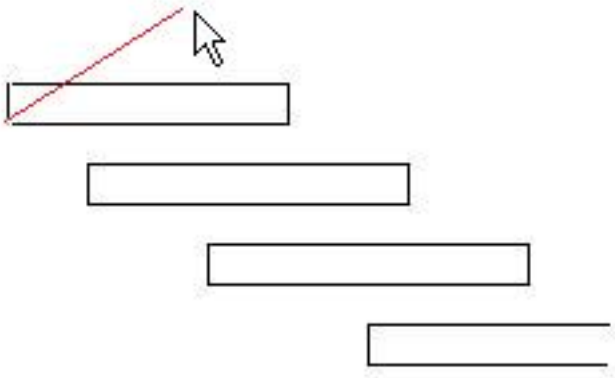


 this icon lets you draw the line.

Set layer and Tool No. in the mode settings bar. Additionally, set "point" on the mouse mode.



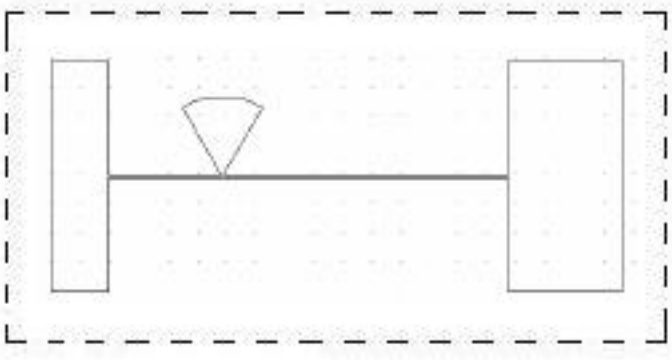
Click on the end points of line to connect with line.
Right click lets you cut the line.



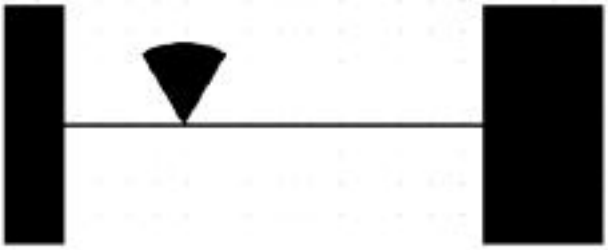
Polygon Fill

Select the closed drawings and then convert the drawings to polygon fill.

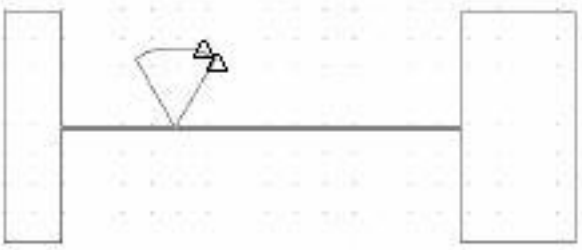
Click on an element or drag rectangle so that element(s) change its color. It is "selected".



 Click this icon to convert the drawings to polygon fill.

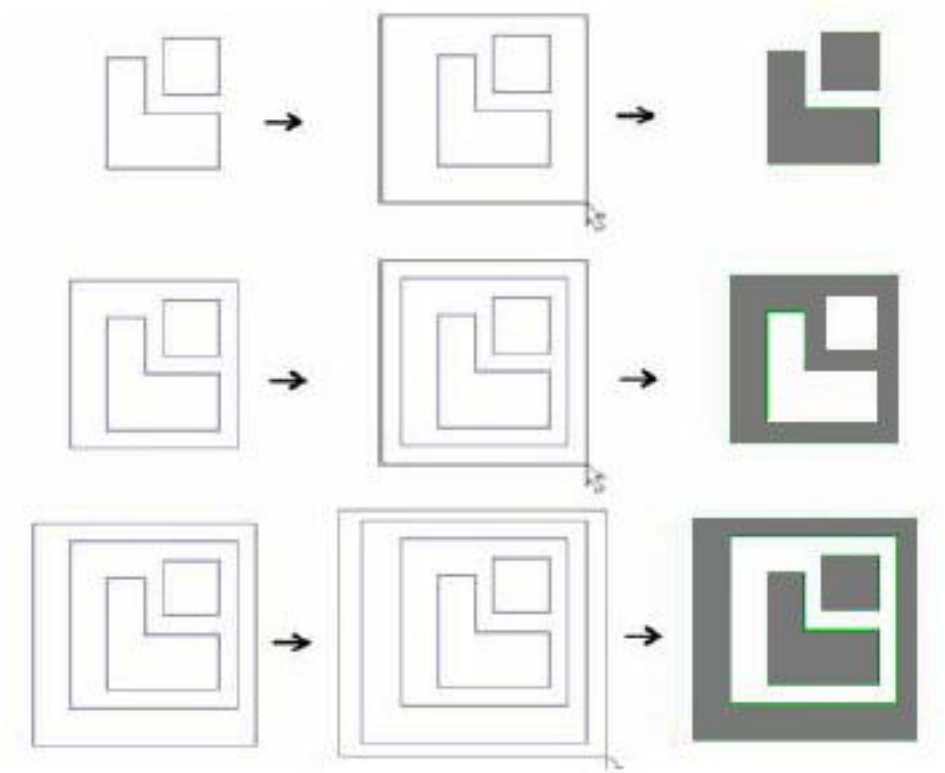


In order to fill, the original drawing needs to be a closed loop.
The program display the triangle mark when the drawing is not closed.
Fix the problem and try to fill again.



Multiple Polygon Fill

When the multiple drawings are selected, the program fill the outside dark, clear the inside, and fill dark again the inside of the inside.



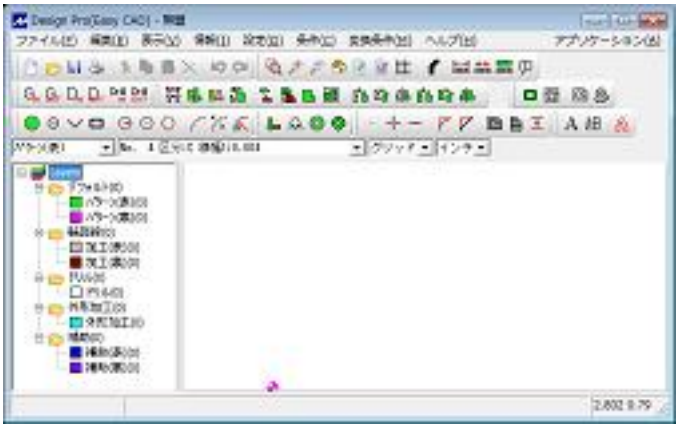
Original Selected Result

Importing Examples

Altium Designer (Protel) RS274X

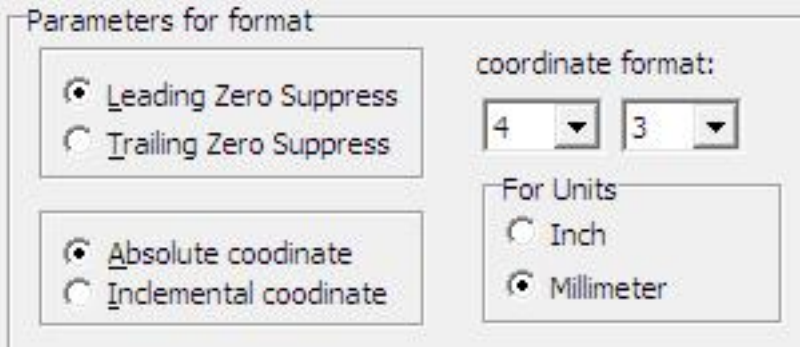
The following sample files are stored in CD-ROM:

- 4port serial interface.GTL (Top Gerber)
- 4port serial interface.GBL (Bottom Gerber)
- 4port serial interface.GM1 (Board Outline Gerber)
- 4port serial interface.TXT (Drill)



- File extension *.GKO is also used for board outline.

Importing Gerber Files :
Drag and drop Gerber files into MITS Design Pro screen and import process is started.
Otherwise, File -> Import -> Gerber In
Select Gerber file and click Open.

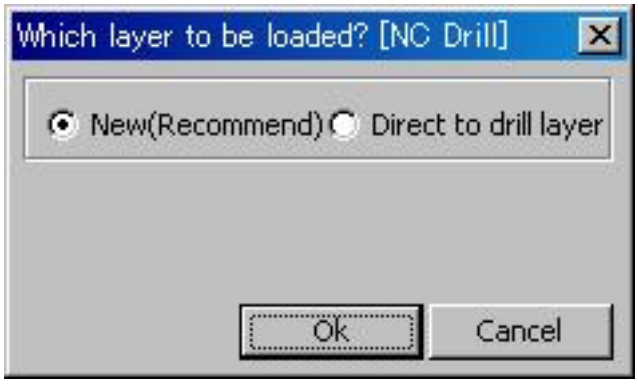


Gerber Settings screen appears.
Not necessary to change. Click Done.

The file will be imported.

Importing Drill file :
Drag and drop NC drill file into MITS Design Pro screen and import process is started.
Otherwise, File -> Import -> Drill In
Select drill file and click Open.

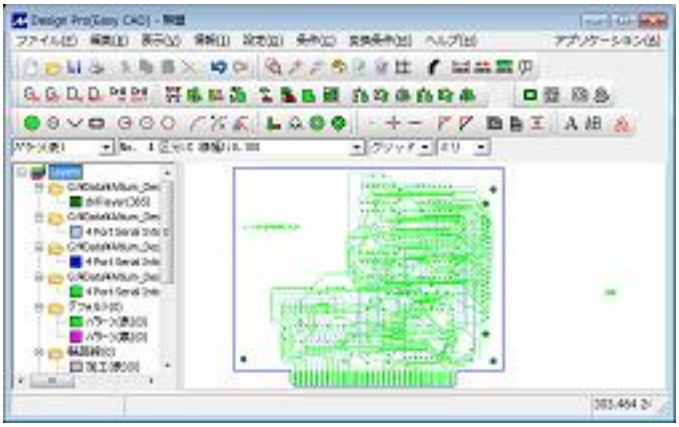
Select New and click OK.



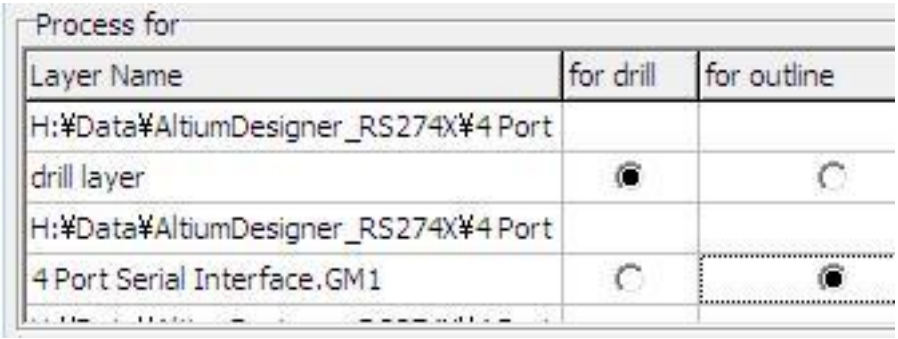
Click Autodetecting button on the NC Drill settings screen.
Choose ..GTL layer for reference and then click OK.
Software will detect the most suitable settings automatically.
Click Done.

The file will be imported.

In the event that the holes are displayed at different location of the patterns.
Autodetecting shifts holes to match pattern but sometimes fails.
In this case, go back to Altium Designer and export Gerber files again without option such as [Center plot on film] or [offset].
Otherwise, you can move holes and align them to the patterns in this screen.



Auto Drill :
Auto Drill re-assigns the tool No. of holes to the standard tool table of the machine.
Click Auto Drill icon



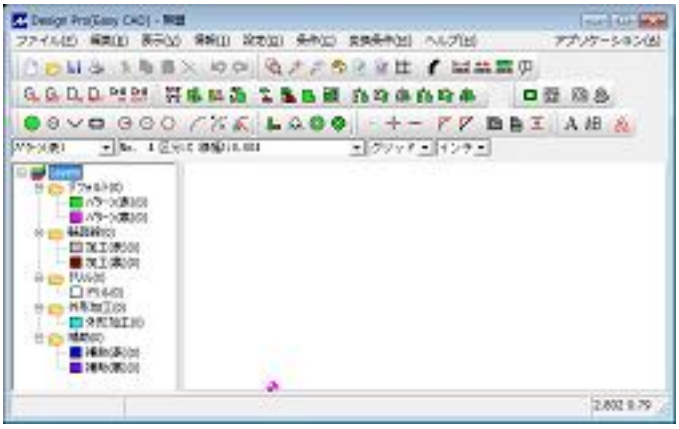
Set Drill layer to "For drill"
And 4port....GM1 to "For outline".
Click Apply.

Auto Drill process will be done and the color of holes will be changed to white.

Cadence OrCAD PCB Editor RS274X

The following sample files are stored in CD-ROM:

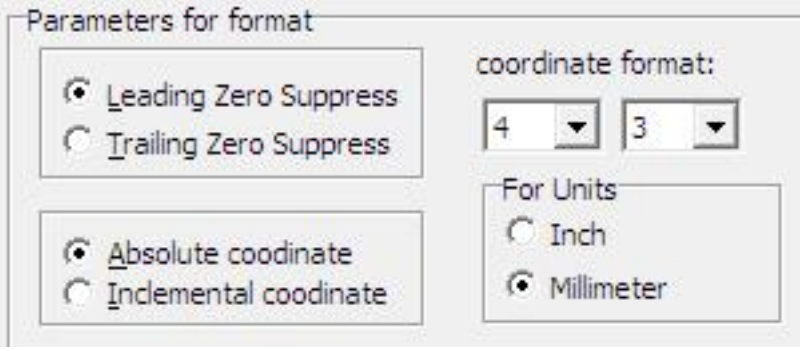
- Top.art (Top Gerber)
- Bottom.art (Bottom Gerber)
- BD_Line.art (Board Outline Gerber)
- sample_R_ALL-1-2.drl (Drill)



Importing Gerber Files :
Drag and drop Gerber files into MITS Design Pro screen and import process is started.
Otherwise, File -> Import -> Gerber In
Select Gerber file and click Open.

Gerber Settings screen appears.
Not necessary to change.
Click Done.

The file will be imported.

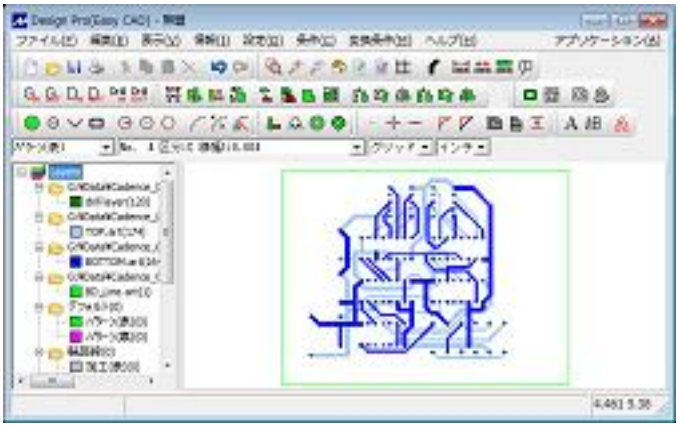


Importing Drill file :
Drag and drop NC drill file into MITS Design Pro screen and import process is started.
Otherwise, File -> Import -> Drill In
Select drill file and click Open.

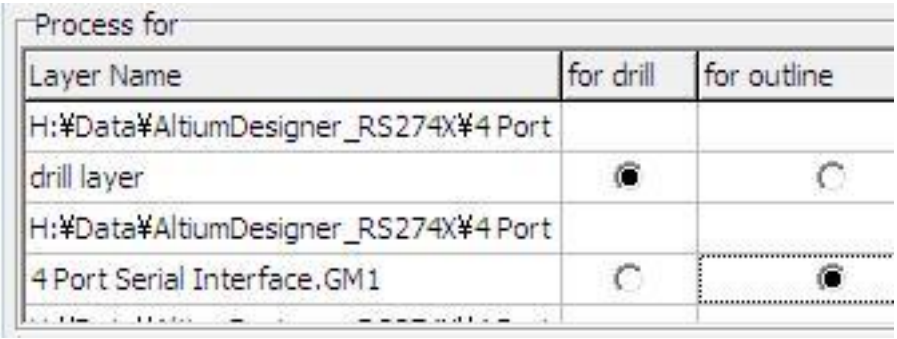
Select New and click OK.

Click Autodetecting button on the NC Drill settings screen.
Choose TOP.ART layer for reference and then click OK.
Software will detect the most suitable settings automatically.
Click Done.

The file will be imported.



Auto Drill :
Auto Drill re-assigns the tool No. of holes to the standard tool table of the machine.
Click Auto Drill icon 



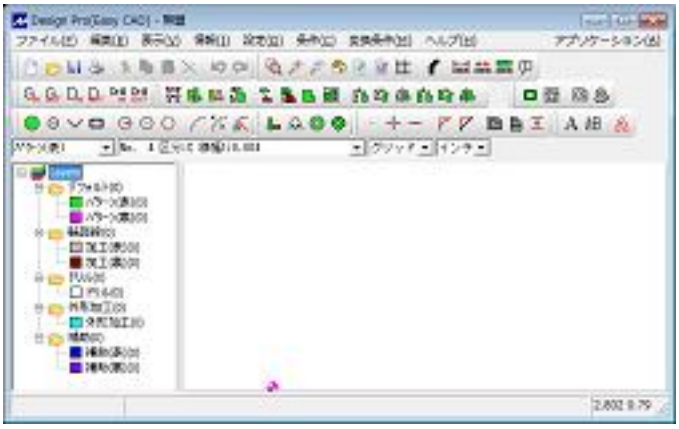
Set Drill layer to "For drill"
And BD_Line.art to "For outline".
Click Apply.

Auto Drill process will be done and the color of holes will be changed to white.

Cadence OrCAD PCB Layout RS274X

The following sample files are stored in CD-ROM:

- Route6lay.top (Top Gerber)
- Route6lay.bot (Bottom Gerber)
- Truhole.tap (Drill)

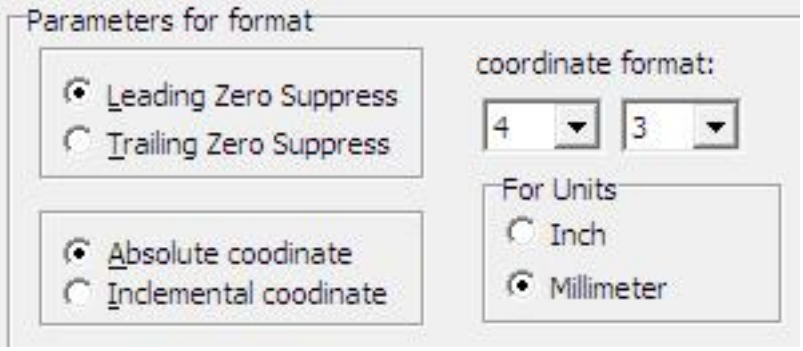


(Sorry, these sample files miss board outline Gerber...)

Importing Gerber Files :
Drag and drop Gerber files into MITS Design Pro screen and import process is started.
Otherwise, File -> Import -> Gerber In
Select Gerber file and click Open.

Gerber Settings screen appears.
Not necessary to change.
Click Done.

The file will be imported.

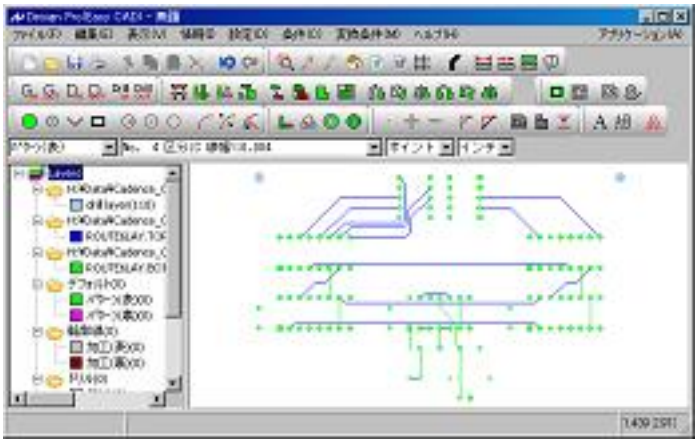


Importing Drill file :
Drag and drop NC drill file into MITS Design Pro screen and import process is started.
Otherwise, File -> Import -> Drill In
Select drill file and click Open.

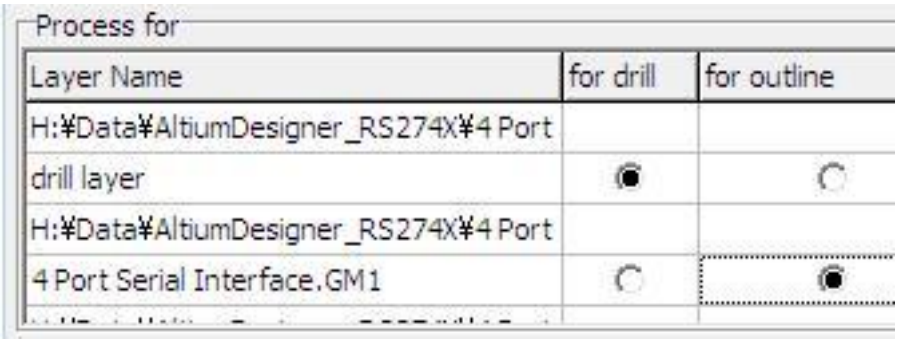
Select New and click OK.

Click Autodetecting button on the NC Drill settings screen.
Choose ...top layer for reference and then click OK.
Software will detect the most suitable settings automatically.
Click Done.

The file will be imported.



Auto Drill :
Auto Drill re-assigns the tool No. of holes to the standard tool table of the machine.
Click Auto Drill icon

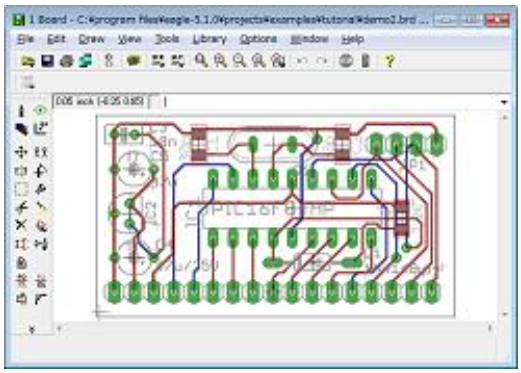


Set Drill layer to "For drill".
Click Apply.

Auto Drill process will be done and the color of holes will be changed to white.

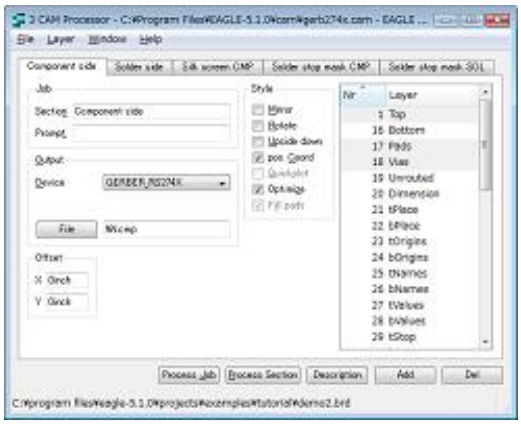
Cadsoft EAGLE RS274X

Sample data : Demo2.brd

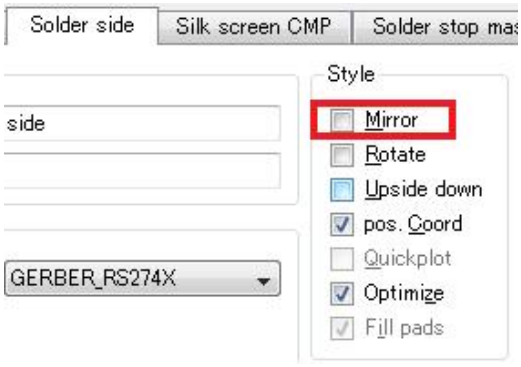


File -> CAM Processor

In CAM Processor screen,
File -> Open -> Job -> gerb274x.cam

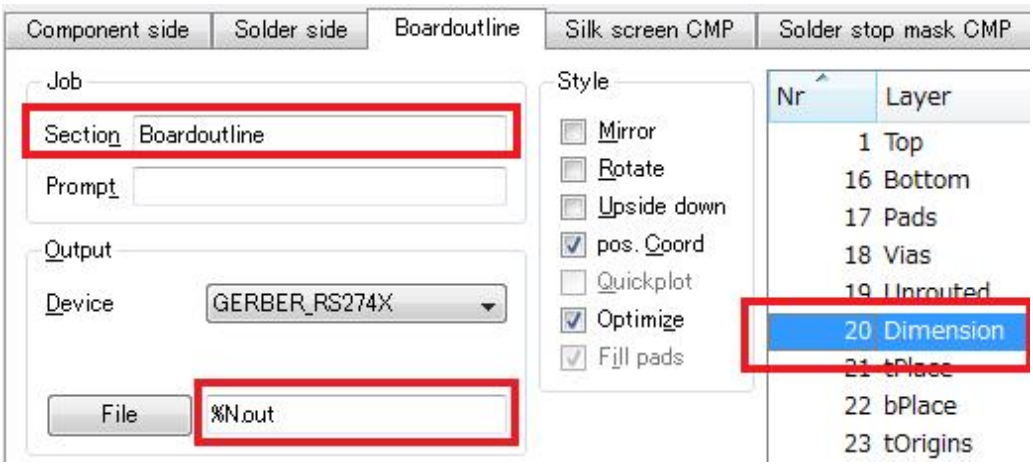


Turn off [Mirror] in Solder Side tab.



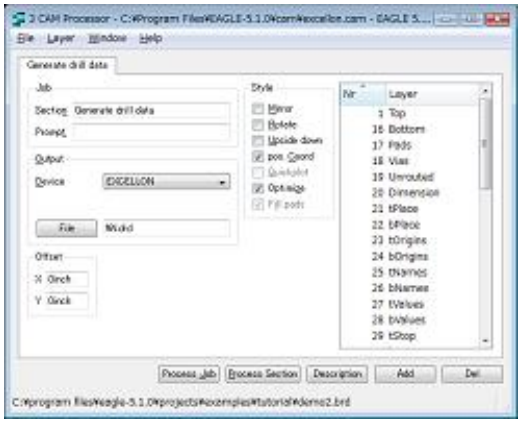
We need to export board outline Gerber.
Click Add at the bottom of screen.
Enter the following settings:

- Enter "Boardoutline" in Section field.
- Enter "*.out" as File extention
- Select only 20 Dimension in layer list.



Click Process Job to export Gerber files.

Next step is export Drill file.
File -> Open -> Job -> excellon.cam



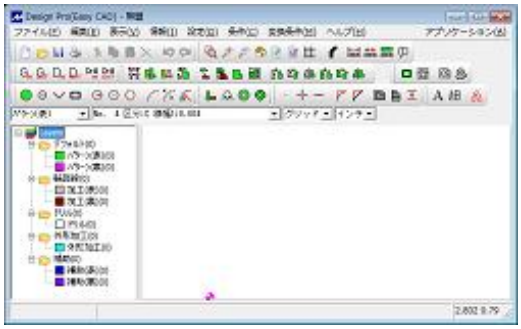
Not necessary to change the settings.
Just click Process Job to export Drill file.

The following sample files are stored in CD-ROM:

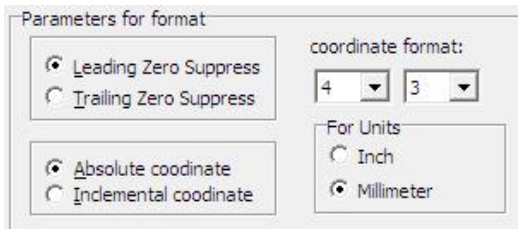
- Demo2.cmp (Top Gerber)
- Demo2.sol (Bottom Gerber)
- Demo2.out (Board Outline Gerber)
- Demo2.drd (Drill)

MITS Design Pro EASYCAD :

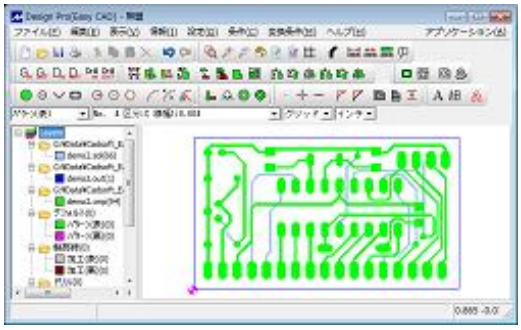
Importing Gerber Files :
Drag and drop Gerber files into MITS Design Pro screen and import process is started.
Otherwise, File -> Import -> Gerber In
Select Gerber file and click Open.



Gerber Settings screen appears.
Not necessary to change. Click Done.

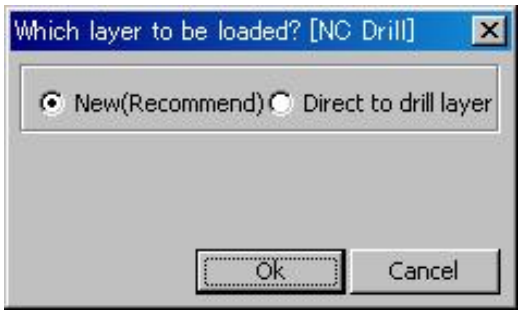


The file will be imported.

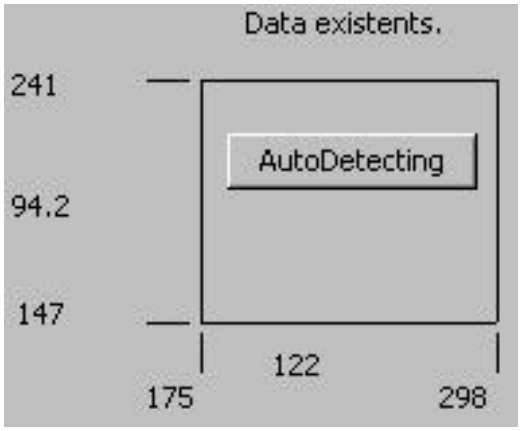


Importing Drill file :
Drag and drop NC drill file into MITS Design Pro screen and import process is started.
Otherwise, File -> Import -> Drill In
Select drill file and click Open.

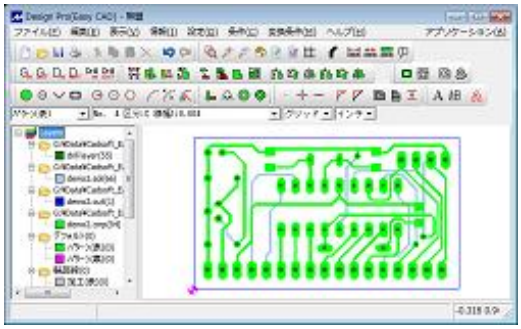
Select New and click OK.



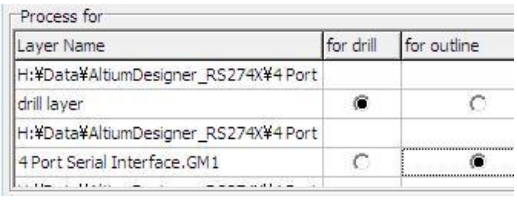
Click Autodetecting button on the NC Drill settings screen.
Choose ...cmp layer for reference and then click OK.
Software will detect the most suitable settings automatically.
Click Done.



The file will be imported.



Auto Drill :
Auto Drill re-assigns the tool No. of holes to the standard tool table of the machine.
Click Auto Drill icon



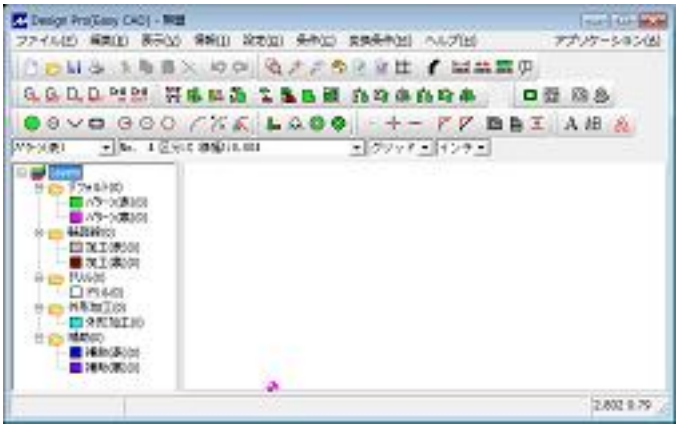
Set Drill layer to "For drill"
And Demo2.out to "For outline".
Click Apply.

Auto Drill process will be done and the color of holes will be changed to white.

CSi WinPCB RS274X

The following sample files are stored in CD-ROM:

- Sample-L1.gbr (Top Gerber)
- Sample-L16.gbr (Bottom Gerber)
- Sample-4.gbr (Board Outline Gerber)
- Excellon.drl (Drill)

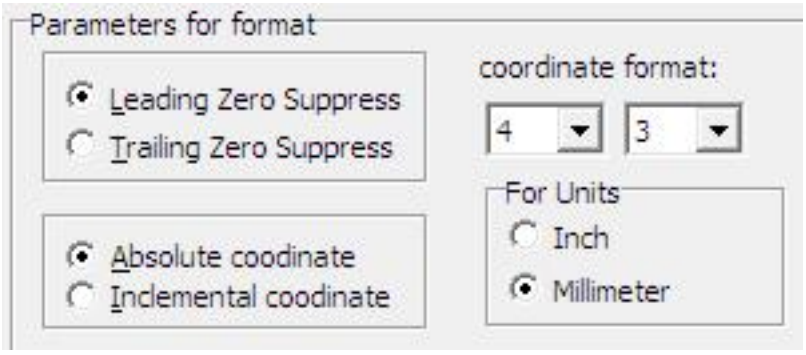


File extention *.nct is also available for drill file.
However, we recommend *.drl because *.drl file contains the information of tool diameter in the header.

Importing Gerber Files :
Drag and drop Gerber files into MITS Design Pro screen and import process is started.
Otherwise, File -> Import -> Gerber In
Select Gerber file and click Open.

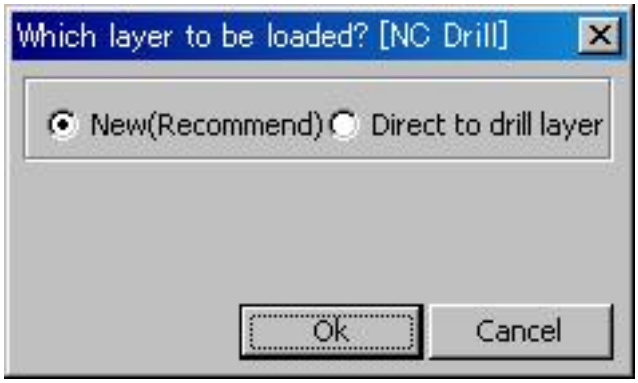
Gerber Settings screen appears.
Not necessary to change.
Click Done.

The file will be imported.



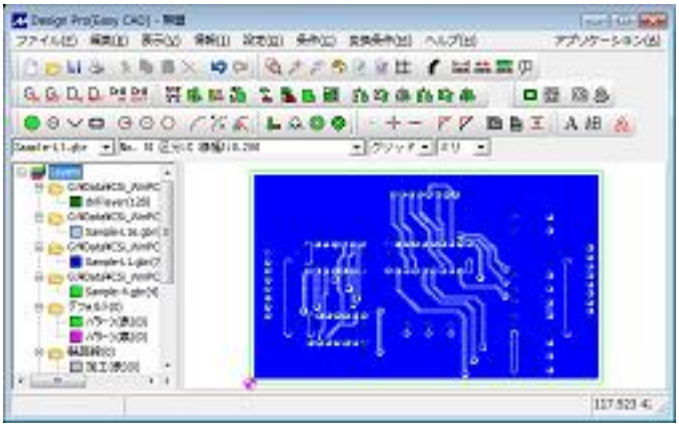
Importing Drill file :
Drag and drop NC drill file into MITS Design Pro screen and import process is started.
Otherwise, File -> Import -> Drill In
Select drill file and click Open.

Select New and click OK.

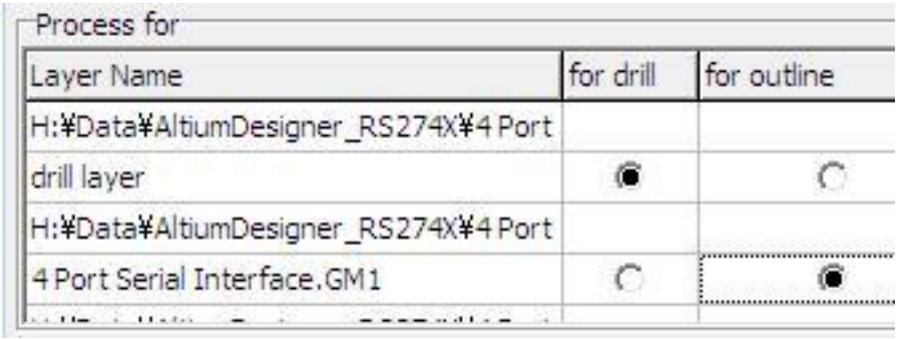


Click Autodetecting button on the NC Drill settings screen.
Choose L1.gbr layer for reference and then click OK.
Software will detect the most suitable settings automatically.
Click Done.

The file will be imported.



Auto Drill :
Auto Drill re-assigns the tool No. of holes to the standard tool table of the machine.
Click Auto Drill icon

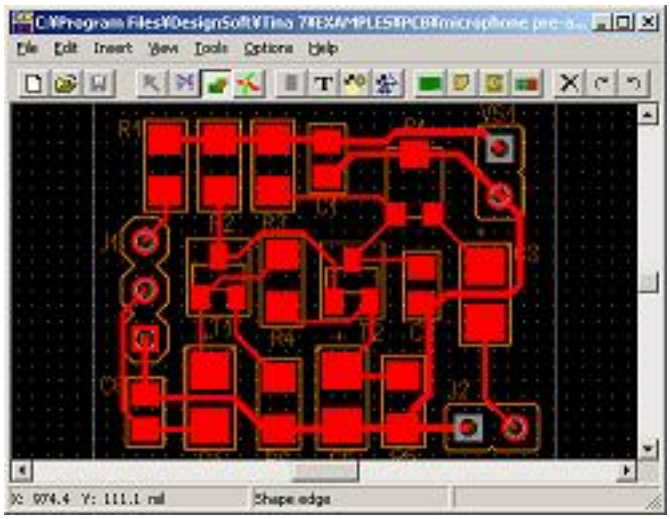


Set Drill layer to "For drill"
And BD_Line.art to "For outline".
Click Apply.

Auto Drill process will be done and the color of holes will be changed to white.

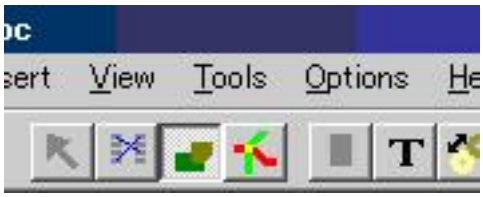
Designsoft TINA RS274X

Sample data :
microphone pre-amp.tpc



Before exporting Gerber files,
we need some settings to export
board outline layer.:

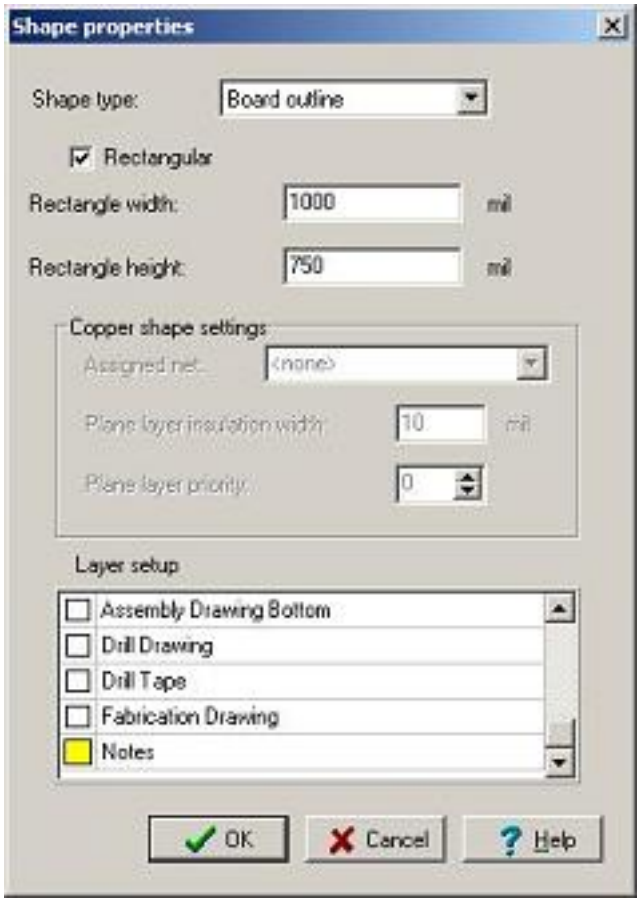
Click Draw/modify shapes.



Double click upon the white line of board outline to call up the screen illustrated at the right side.

Double-click "Notes" layer
in the list.

This setting will export board outline separating from top or bottome Gerber.
It will reduce some operations after importing files in Mits Design Pro.



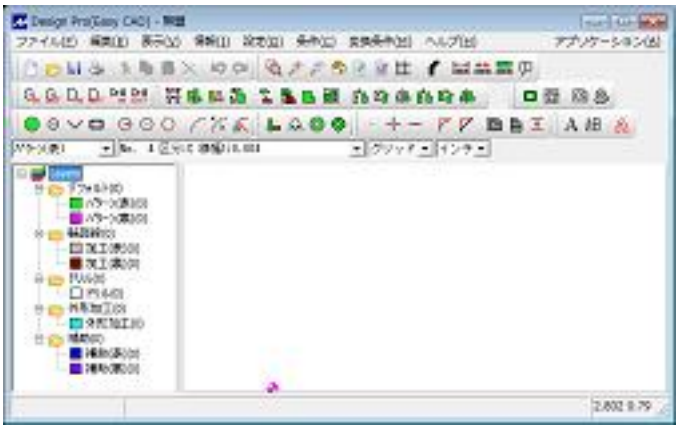
Choose File → Export gerber file.
Enter File name and click save.

The following sample files are stored in CD-ROM:

- Microphone pre-amp.TOP (Top Gerber)
- Microphone pre-amp.NOTES (Bottom Gerber)
- Microphone pre-amp.drl (Drill)

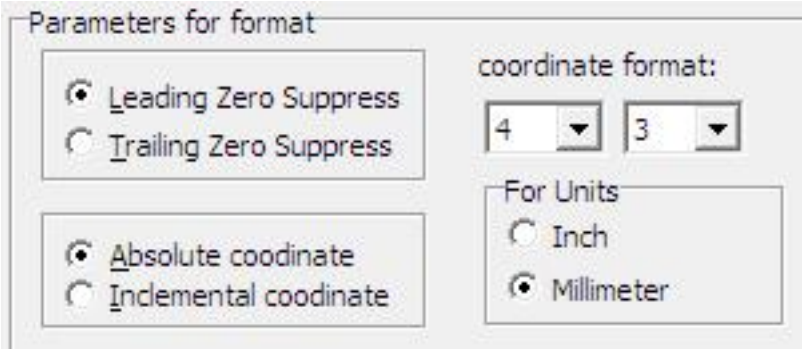
- Bottom Gerber file has the extention *.BOTTOM.

MITS Design Pro EASYCAD :



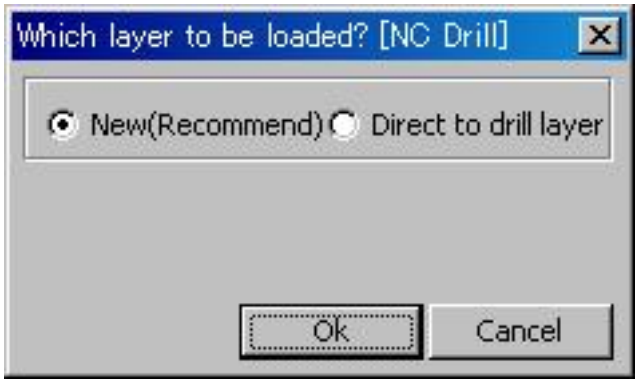
Importing Gerber Files :
Drag and drop Gerber files into MITS Design Pro screen and import process is started.
Otherwise, File -> Import
-> Gerber In
Select Gerber file and click Open.

Gerber Settings screen appears.
Not necessary to change.
Click Done.



The file will be imported.

Importing Drill file :
Drag and drop NC drill file into MITS Design Pro screen and import process is started.
Otherwise, File -> Import -> Drill In
Select drill file and click Open.

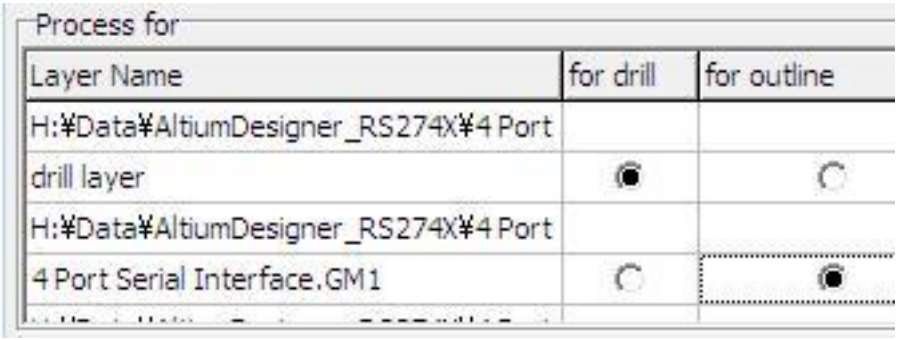


Select New and click OK.

Click Autodetecting button on the NC Drill settings screen.
Choose ...TOP layer for reference and then click OK.
Software will detect the most suitable settings automatically.
Click Done.

The file will be imported.

Auto Drill :
Auto Drill re-assigns the tool No. of holes to the standard tool table of the machine.
Click Auto Drill icon



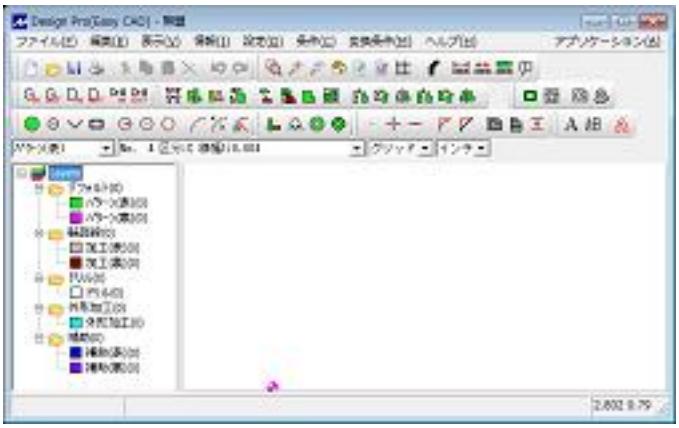
Set Drill layer to "For drill".
Set mirocophone..NOTES to "For outline".
Click Apply.

Auto Drill process will be done and the color of holes will be changed to white.

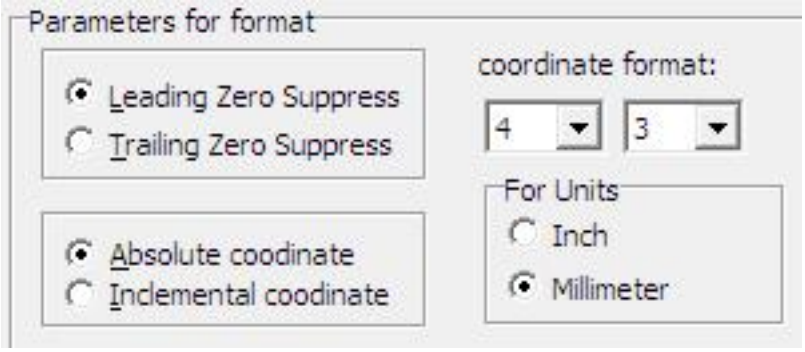
Mentor Graphics PADS RS274X

We have a lot of experience of importing Gerber data and Drill data exported from PADS.

However, we don't have sample files to show the customers.



Importing Gerber Files :
Drag and drop Gerber files into MITS Design Pro screen and import process is started.
Otherwise, File -> Import -> Gerber In
Select Gerber file and click Open.



Gerber Settings screen appears.
Not necessary to change. Click Done.

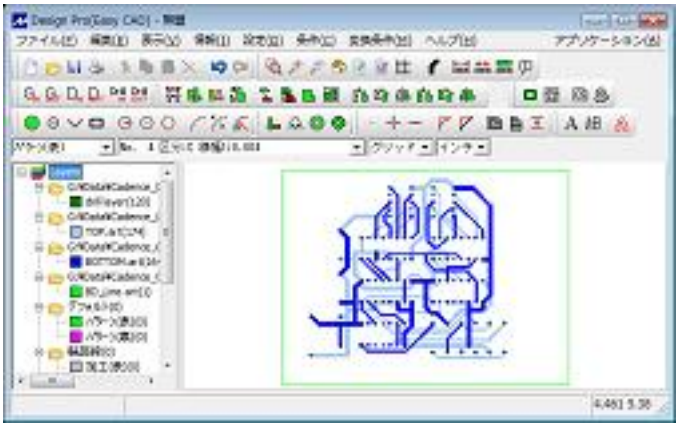
The file will be imported.

Importing Drill file :
Drag and drop NC drill file into MITS Design Pro screen and import process is started.
Otherwise, File -> Import -> Drill In
Select drill file and click Open.

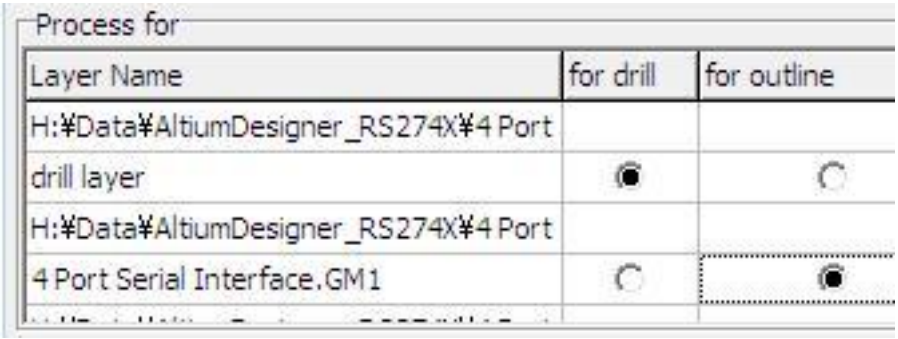
Select New and click OK.

Click Autodetecting button on the NC Drill settings screen.
Choose Gerber layer for reference and then click OK.
Software will detect the most suitable settings automatically.
Click Done.

The file will be imported.



Auto Drill :
Auto Drill re-assigns the tool No. of holes to the standard tool table of the machine.
Click Auto Drill icon



Set Drill layer to "For drill"
And Boardoutline layer to "For outline".
Click Apply.

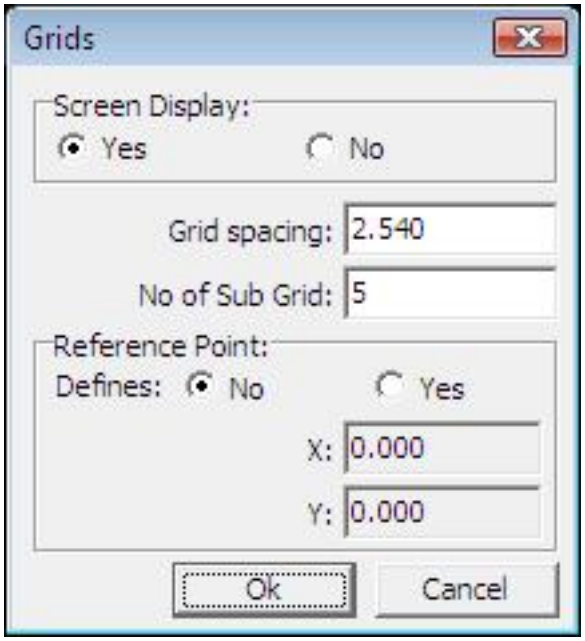
Auto Drill process will be done and the color of holes will be changed to white.

Drawing with EASYCAD

Grid Settings

Grid helps the operation such as drawing line, pad and parts.
It is convenient to set grid pitch same as IC pitch.

Choose Work Prefs. -> Grid.

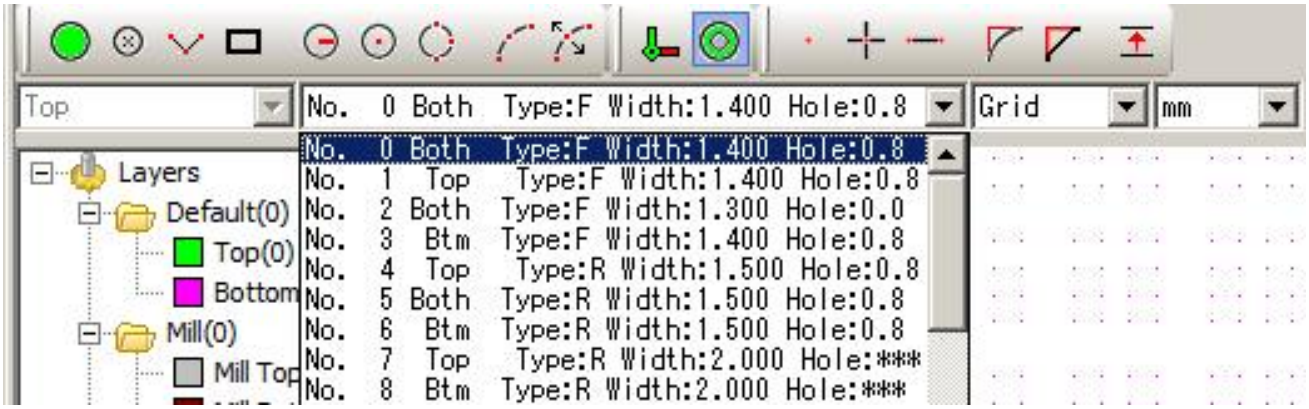


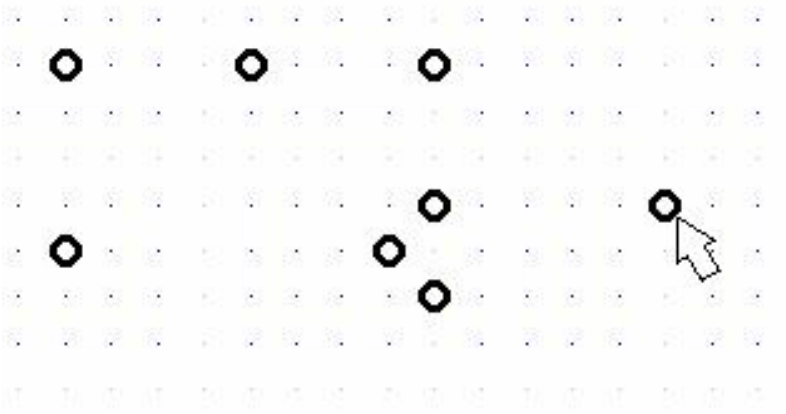
Enter grid pitch.

Pad

This icon produces a pad on the screen.

Choose Grid in the mode settings bar.
And also, choose the desired pad to be drawn in the mode setting bar.





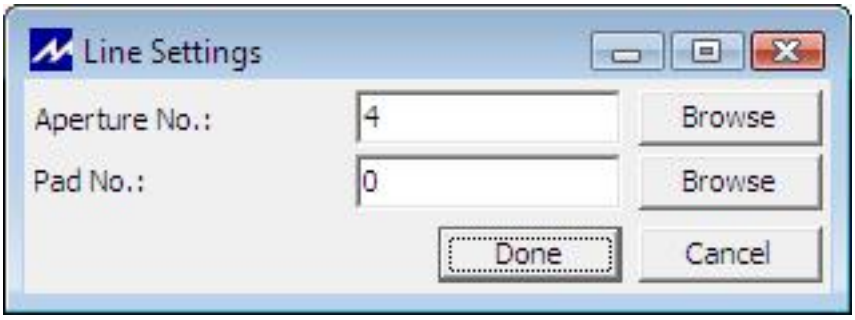
Specify the point for a pad.
Click on the location to draw the pad.

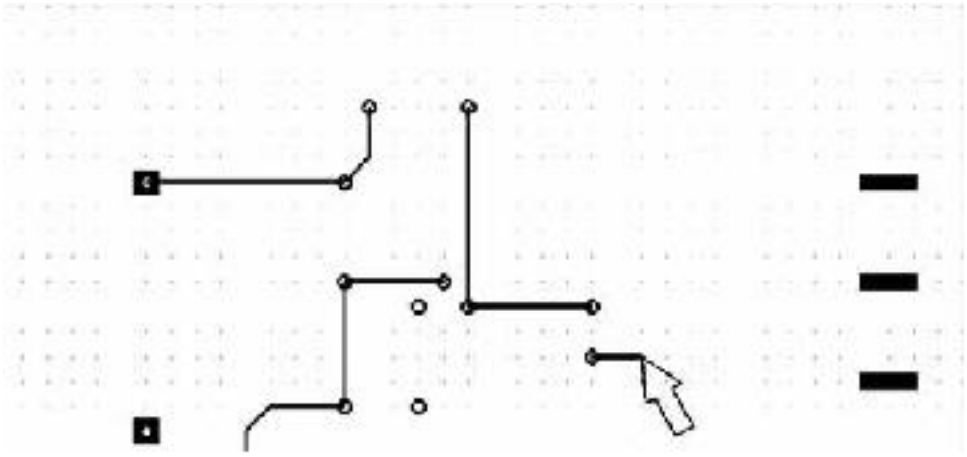
NOTE:
Pads can be drawn ONLY in Default(Top)(Bottom) and Drill layer.

Line & Pad

 This icon lets you draw line & pad.

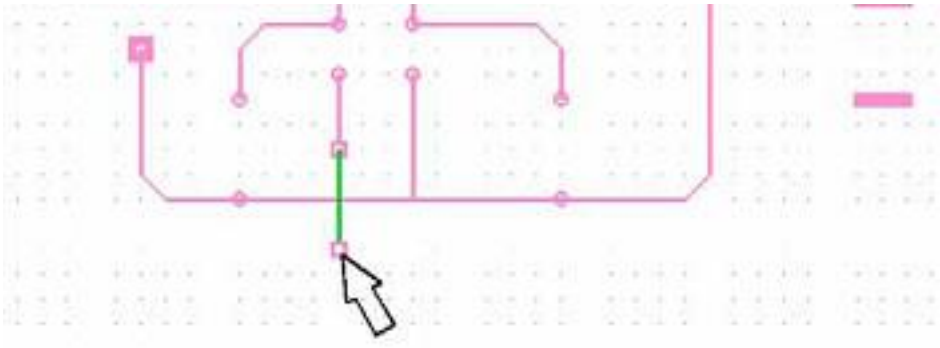
When the icon is clicked, the Line Settings screen will appears.
Click Browse to select the line and pad you want to use.





Specify the routing start point (Top)
Click on the start point.

Specify the routing path (Top)
You can draw a line for the pattern on the top side.
Otherwise, double-click right after clicking on the start point.
And the message will change **the routing path (Bottom)** and you can draw a line on the bottom side.
Right click lets you cut the line.
The drawings are displayed from Top side view, it means the drawing on the bottom side is displayed mirrored on screen.



Double click lets you switch from Top side to Bottom side (or reverse way).
The pad that was selected in the Line Settings screen will automatically appear.

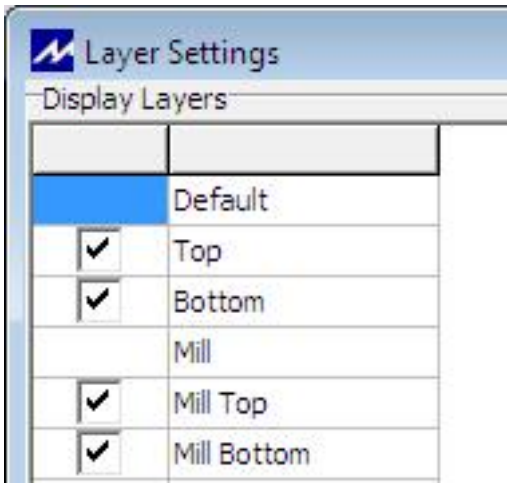
NOTE:
This command lets you draw the line and pad ONLY in Default layer (Top) (Bottom) and Drill layer.

Print

Prints out PCB data to the printer or the plotter.

Choose File -> Print.

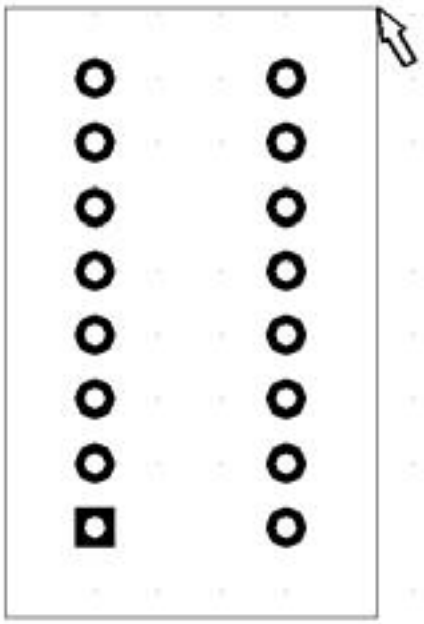
Print size:
Choose whether program adjusts the print size according to the paper size or you specify the rate by yourself.



Print Layers:
Check layers to be printted out.

Save Part

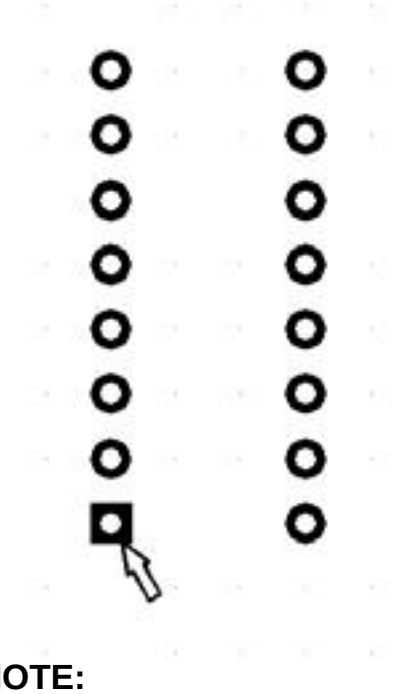
It is convenient to save the drawing as part data which is frequently used.



Click on an element or drag rectangle so that element(s) change its color.
It is selected.

Choose File -> Save Part.

Enter a name of part file and then Click SAVE.
Leave it blank if items in File Information screen are not needed.



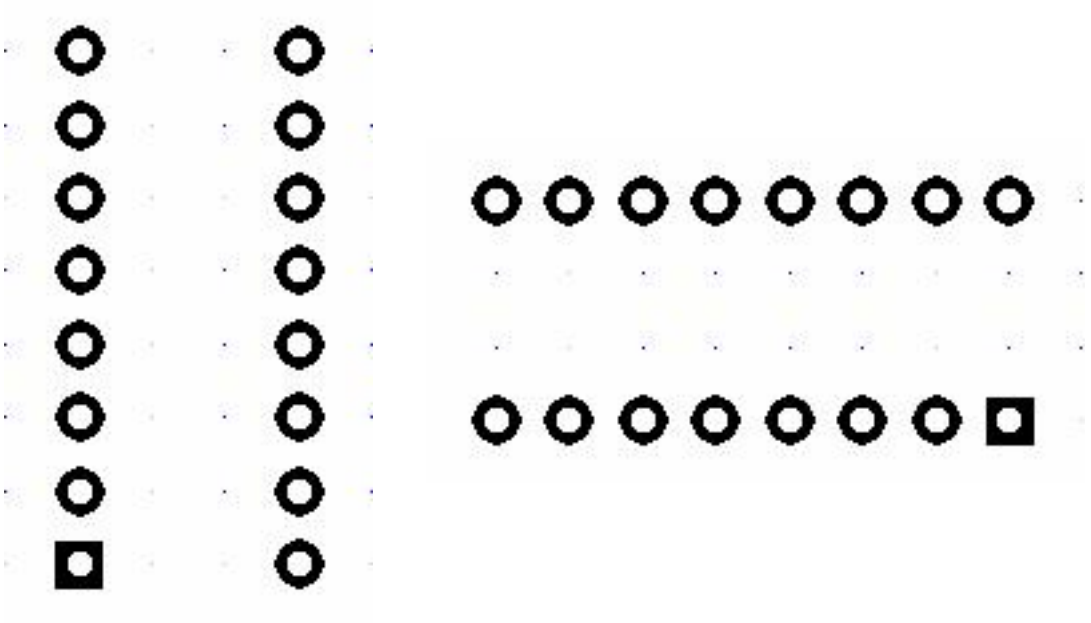
Specify the reference point
Click the point on the screen you want to use as the reference point, or specify the point by entering its coordinates.

NOTE:
ONLY the data in the following layers can be saved as part data.:
Default(Top)(Bottom), Mill(Top)(Bottom), Drill, Routing, Subsidery(Top)(Bottom)

Place Part

Lets you select part data that has already been saved and place it at a specified location

Choose File -> Place Part.
Select the part data file you want to use.
In Enter Angle screen, enter the angle at which the part is to be placed.
The entered angle rotates the part data counterclockwise on the center of the home point of the part data.



0 degree 90 degree

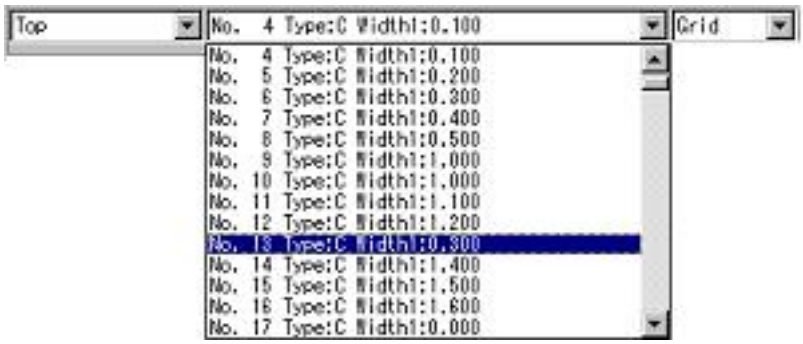
Specify the placement point.
Click the point on the screen where you want to place the part, or specify the point by entering its coordinates.

Change Attribute

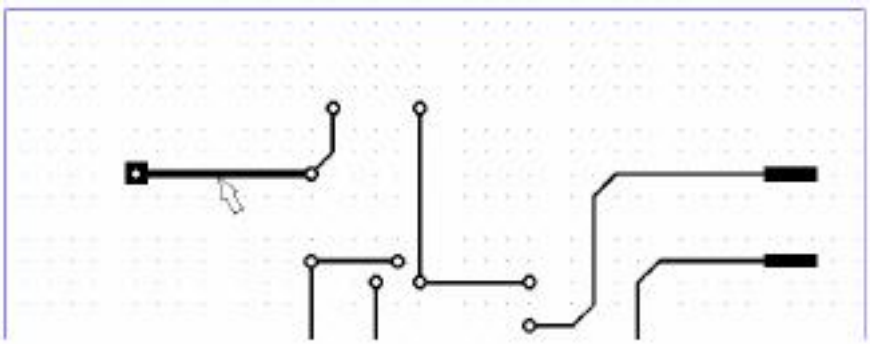
Change Line Width - 1

 This icon lets you change the layer attribute(s) of a specified group.

*  Change attribute one by one.
*  Change attribute of a specified group.



Set mode settings bar to indicate the layer and tool where the element is to go.

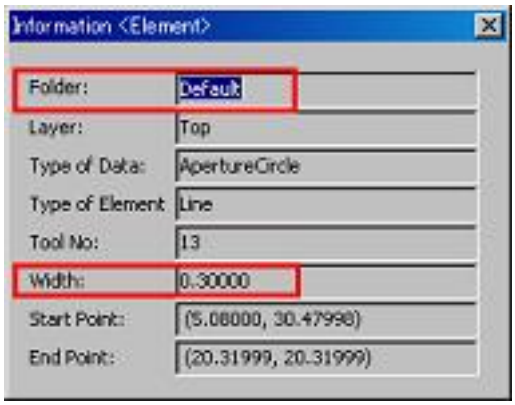


[Select an element] message appears in status bar.
Click on the element to be changed.

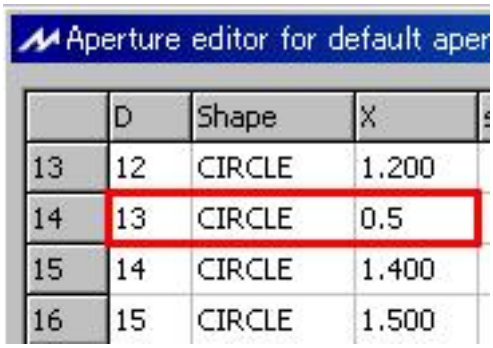
Change Line Width - 2

When you want to change all the lines with 0.3 mm width to 0.5 mm width, first you can click and confirm the information of the element using

 command.



[Select an element] message appears in status bar.
Click on the element to be changed.



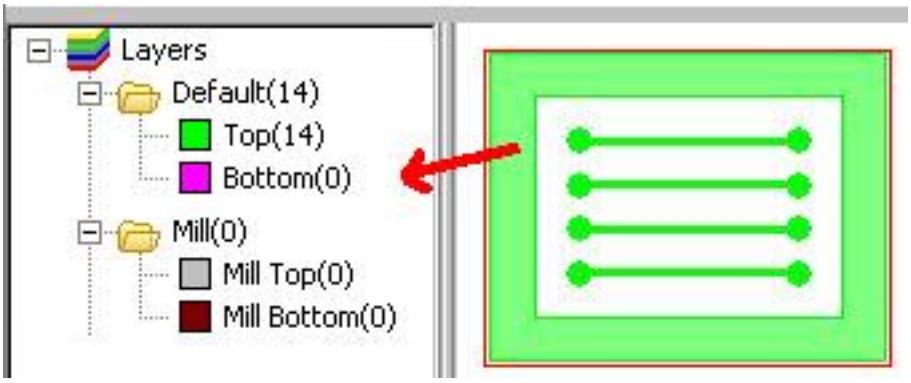
Next, Right click -> properties on the folder to call up the aperture editor screen.
Change the width 0.3 to 0.5 on D13. All the D13 data will change its size to 0.5mm.

Move Layer from Top to Bottom

When you want to move the drawings on Top layer to Bottom layer....

For example, draw something on Top Layer on Default folder.

Select the drawings and drag'n drop on Bottom layer.
The data will be moved to Bottom layer.

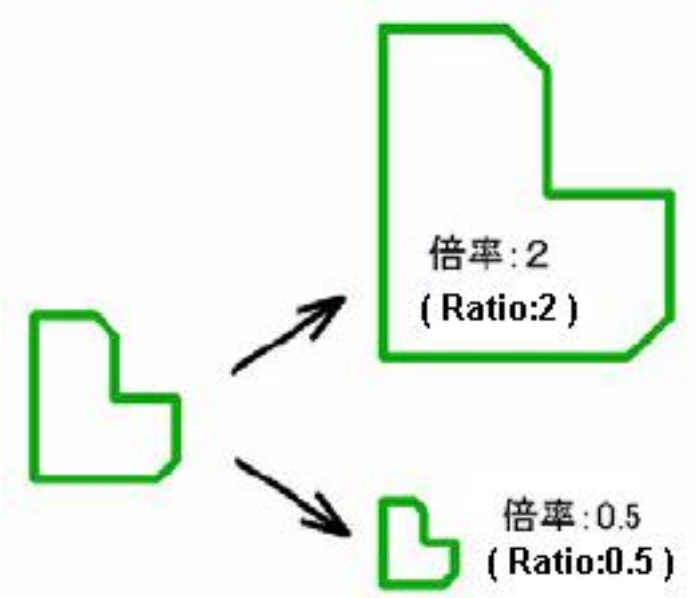


Enlarge

When you want to change the size of drawings...

Select the drawings and click the icon 

Enter Ratio and click on the reference point to change the size of drawings.
Take care that the distances or lengths are to be changed but the tool size are not to be changed.



Fonts

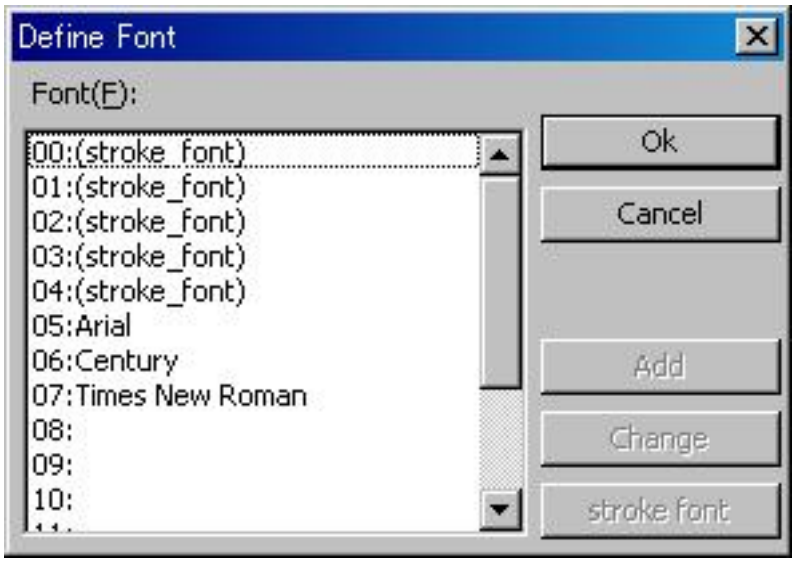
This software allows you to use the following 2 type of fonts:

1. Stroke fonts
   It is consists of center lines of character.
   Alphabet and numeric characters only.

2. Outline fonts
   It supports TrueType fonts.

It requires the following settings in advance to use fonts.

Choose Work Prefs. -> Fonts and then register font in the screen.



Click any line in the list and it will turn blue.
If you want to register stroke font, click [stroke font] button in the right of the sreen.

If you want to register truetype font, click [Add]/ [Change] button to display the list of truetype fonts.
And then choose any fonts you like.

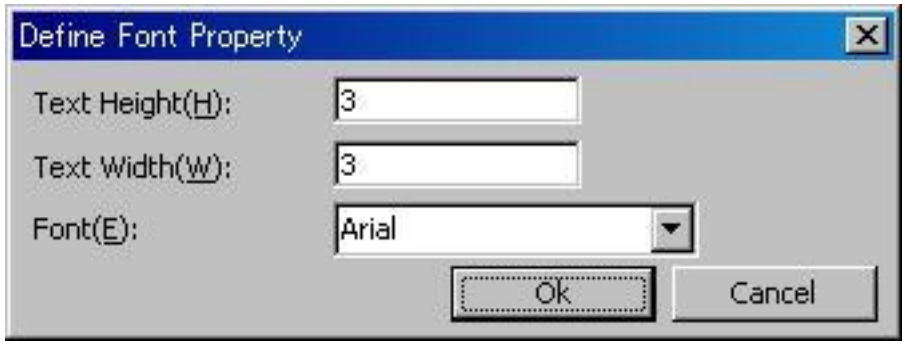
In order to import Flash for Windows data, it is recommended to set this screen same as Flashwin settings. Please refer to Work Prefs. -> Fonts in Flashwin menu.

Close the Font settings.

Choose Work Prefs. -> Text.

Choose any font you like in the font list and enter height and width of character.

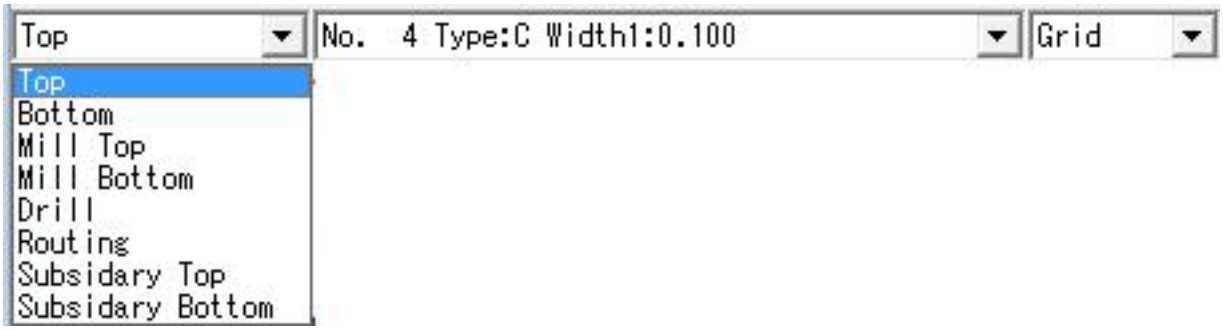
These are default settings of text when you draw and import DXF file.



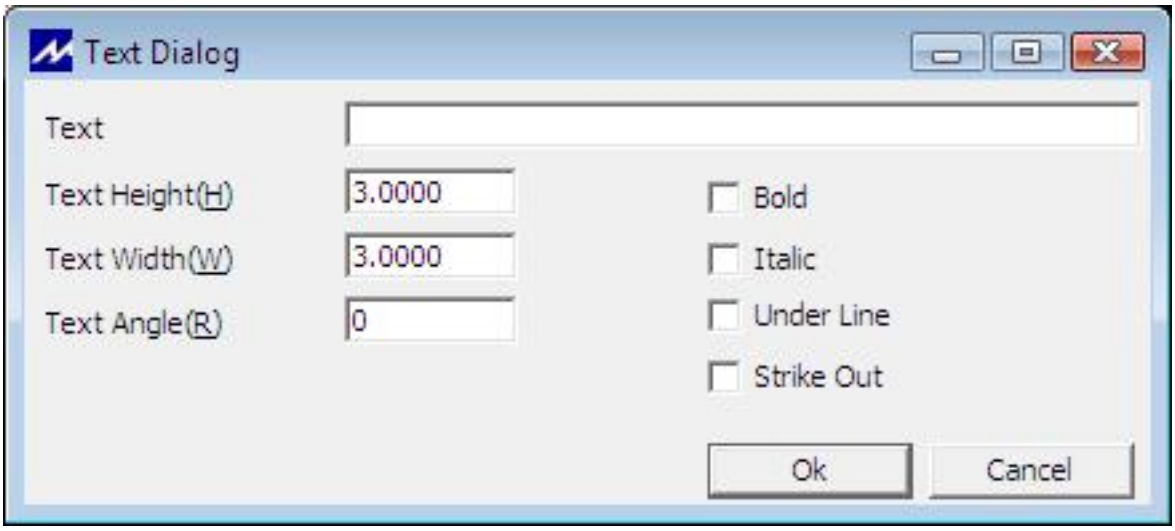
Input Text

A This icon lets you produce character string data.

Set the layer for the new text data on the Mode Setting Bar.



Specify location for comment.
Click on the location where the character string is to be produced, or enter its coordinates.



Input comment and other settings in the dialogue.

The following parameters are invalid when text is written using stroke font:
Bold, Italic, Under Line, Strike Out

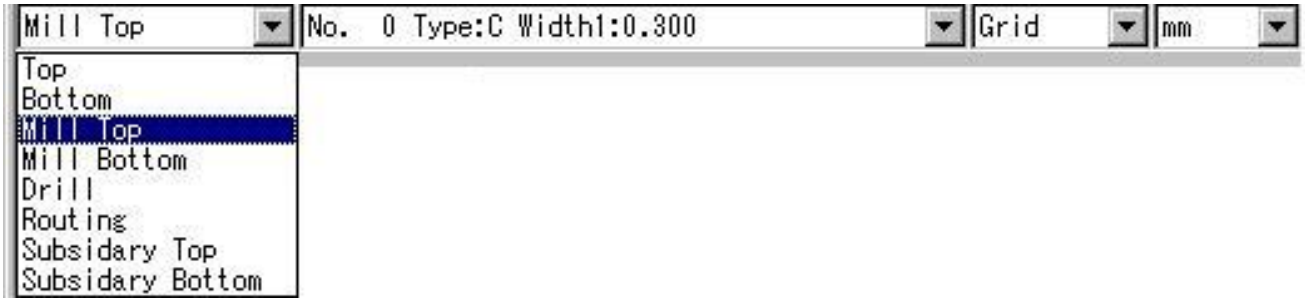
Text to Milling Data

Text string cannot be treated as line segment or arc.
It is necessary to de-assemble the text string to lines or arcs in order to mill text.

 This icon lets you put a line segment or arc in place of text string.

1. Text with Stroke fonts:

Choose Mill(Top) or (Bottom) layer in the mode settings bar. Also choose any tool diameter in the list box on the right side of the layer list box.



Click deassemble text icon and click on the text to be milled.
It is converted to lines and arcs and moved to mill layer.

2. Text with Outline fonts:

Click deassemble text icon and click on the text.
The text string is converted to the polygon data in the same layer where it used to be.

When you generate milling data on this layer, the milling data will be generated around the text string same as other pattern data.

Generating Milling data

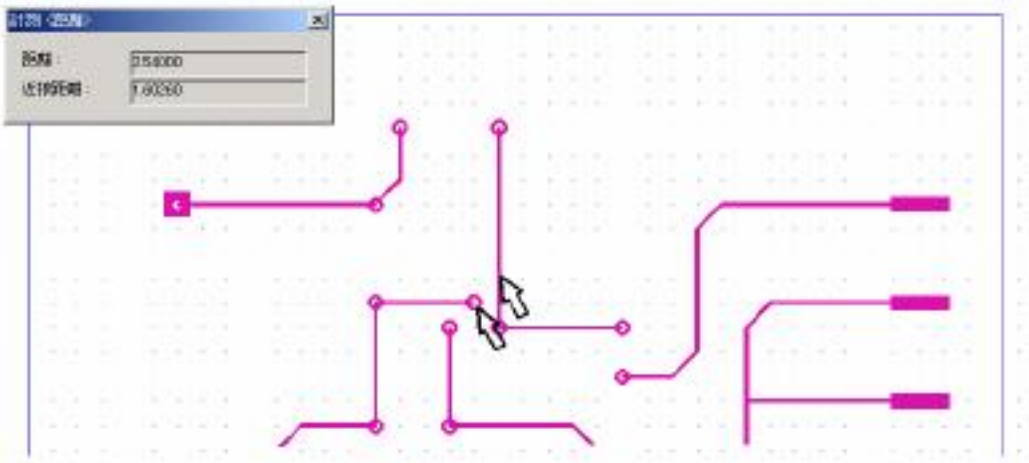
Measure Distance

Measure the minimum gap between 2 elements in the designed pattern.
It will help to decide what diameter is recommended when generating milling outlines.

 This icon lets you measure the distance of 2 elements.

Click on 2 elements to be measured. And the display returns 2 values.

Distance:	between the centers of the 2 elements.
Closest:	between their nearest points.

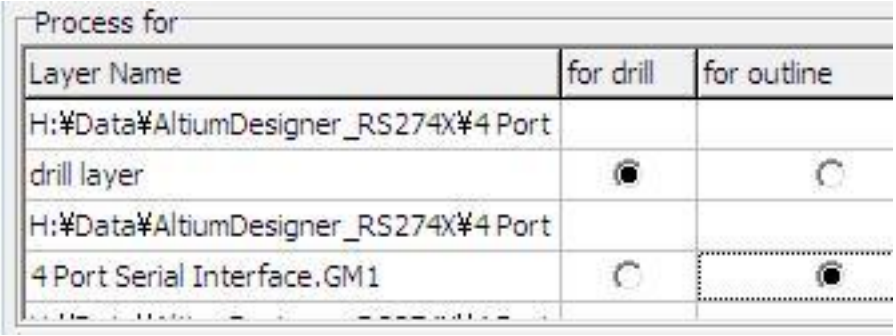


Auto Drill

Auto Drill re-assigns the tool No. of drill imported to the standard tool table of the machine.
Some holes with small diameter are to be sorted and merged by size.
Some holes with large diameter are to be moved to board outline layer in order to be process by contour routing.
Additionally, circles in DXF data which are drawn at the locations of holes are also processed by Auto Drill.

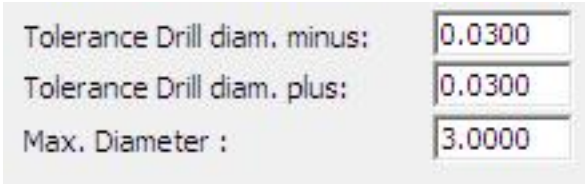
 Click this icon.

Layer:
Set layer to "For drill". Flash and Circle data in this layer will be converted to the drill data and stored in the drill layer.



Set layer to "For outline". Some data with large diameter will be moved into this layer.

Tolerance:
For example, 0.032 inch is 0.812 mm in metric unit. 0.040 inch is 1.016 mm in metric unit.
Tolerance is necessary to sort and merge the drill diameter with small integer value.



For example again, there are 0.8mm and 1.0mm in the drill table.
When 0.030mm is set in the minus and plus tolerance field, some circles and flashes between 0.770 and 0.830 in metric unit will be merged to 0.8mm drill data.
Some between 0.970 and 1.030 mm diameter will be merged into 1.0 mm drill data.

Max Diameter:
When the Max Diameter is set 3.0mm, some circles and flashes with bigger than 3.0mm diameter will be moved to the layer set as "For outline".

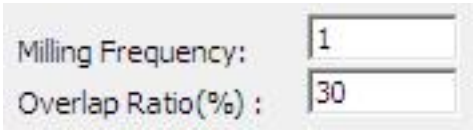
Generating Milling Outlines

 This icon let you generate milling outlines.

Layer:
Set "Layer Type" to "Pattern Top". The program will generate the milling outline on the Top side. The outline will be stored in "Milling Top" layer.

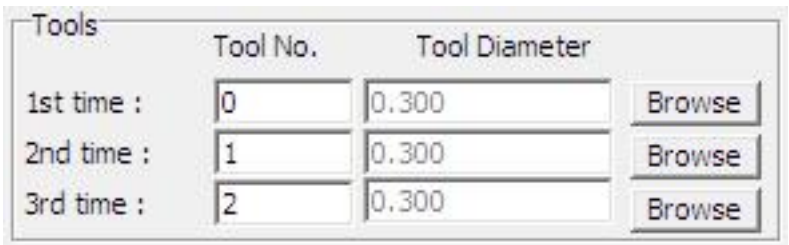
Set "Layer Type" to "Pattern Bottom". The program will generate the milling outline on the Bottom side. The outline will be stored in "Milling Bottom" layer.

Set "Layer Type" to "PCB outline". This definition is used by the rubout and routing generation
which is described later.



Milling Frequency:
The larger this number is, the wider the milling width and easier it will be for solder to be applied.
But time required for data processing and milling will be increased accordingly. Therefore, 2 or 3 times is recommended.

Overlap Ratio(%):
This value can adjust the overlap ratio when repeating is more than 2 times. The recommended value is 30%. However, it depends on the milling width and the tool bit shape. Small ratio could produce the remain of copper.

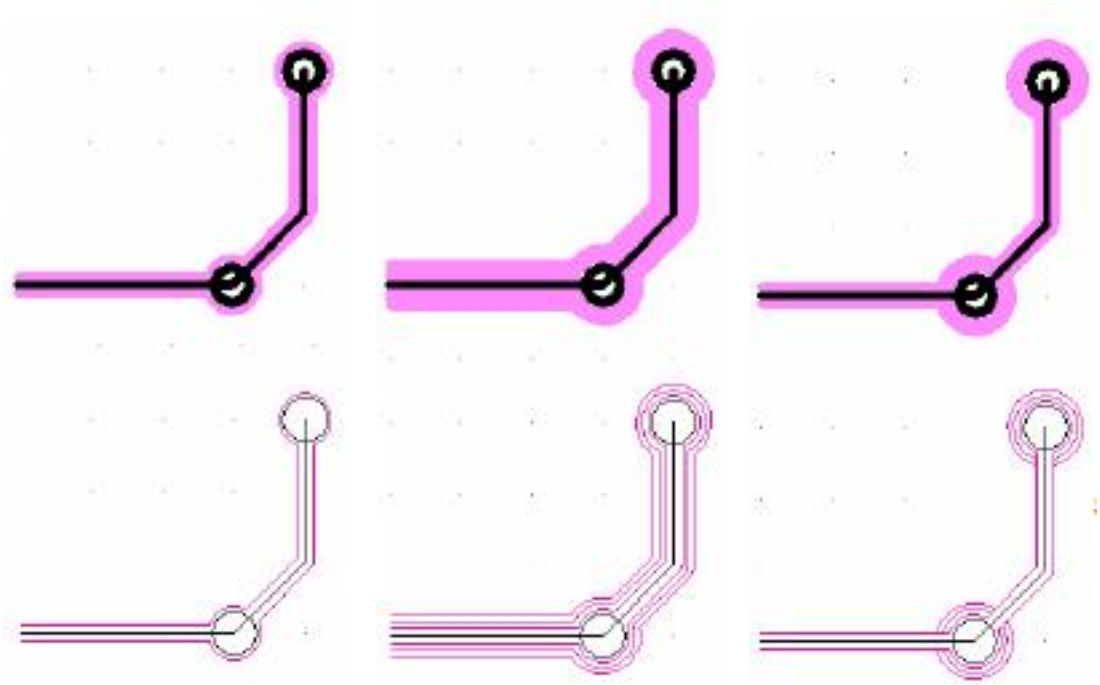


Tool No.:
Click Browse to select the tool No. (tool diameter = milling width)
* Select the tool diameter (milling width) smaller than the minimum gap of the patterns.
If the milling times is more than once, enter the tool No. for the 2nd and 3rd times.
* Take care that the milling will be processed from smaller No. of tool, not from smaller diameter of tool.

Mill only pads from 2nd time:
When Milling Frequency is set to more than 2 times and this option is checked, milling outline will be repeated only on pads.
Pads means data whose attribute is Flash.

Difference of milling options:

Outline 1 time	Outline 3 times	3 times and mill only pads

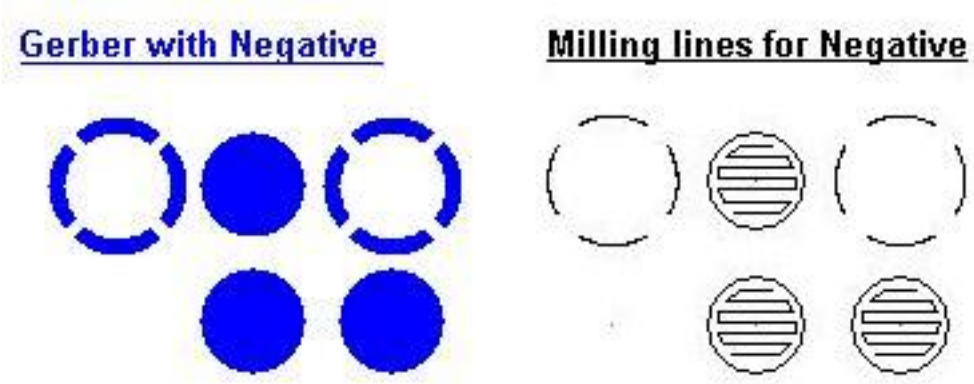


(Center line display)

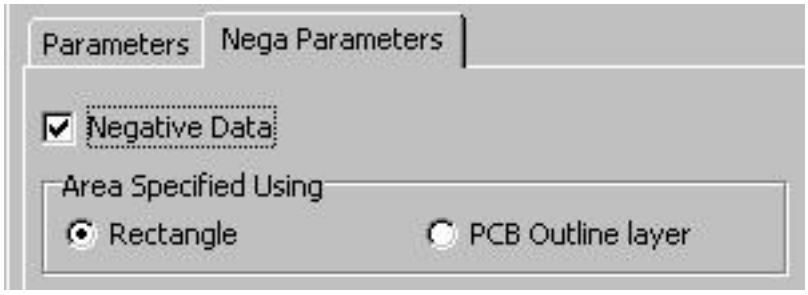
 This icon lets you change the display of width: Fill, Center Line, Ball&Stick

Generating Milling Outlines (Negative)

Negative Data:
When you want to make a multilayer board
and the gerber files of internal layer are exported negative polarity,
the milling outlines needs to be generated with inverse direction.

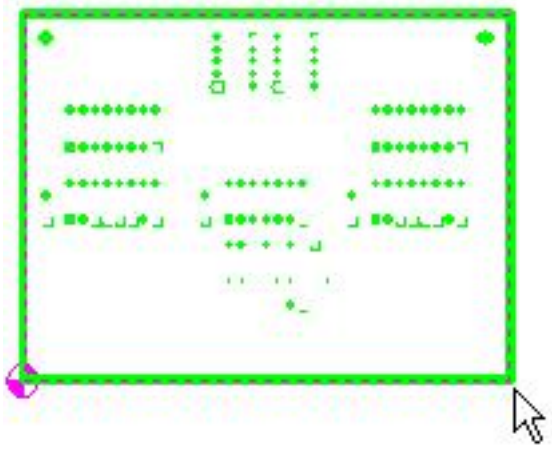


In order to generate milling outlines for negative gerber,
check [Negative Data] in [Nega Parameters] tab.



And also, it needs to select how to serect rubout area at this time, whatever you want rubout or not.
Select [Rectangle] or [PCB outline layer] for specifying area.

When Rectangle is selected as rubout area,
click [Apply] and select area.
Program message will invite you to specify area,
first "Specify first corner" and then "Specify opposite corner".

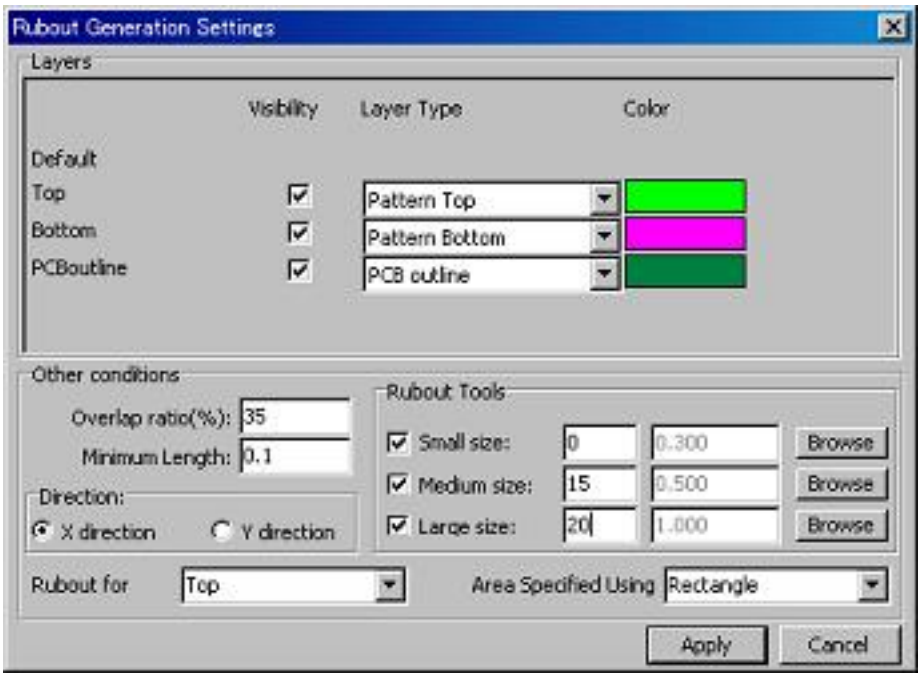


When PCB outline layer is selected as rubout area,
program will seach the closed drawing in the layer which is specified as layer type "PCB outline".
And the closed drawing found will be applied as rubout area.

Generating Rubout Data

This command generates rubout data for patterns in which milling outlines have been generated.
Milling outlines needs to have already been produced in order to generate rubout data.

 This icon lets you generate rubout data.



Rubout for:
Select Top, Bottom or Both side on which rubout data is generated.

Rubout Tools:
Large area is milled with large tool and small area is milled with small tool.
Check the tool to be used.

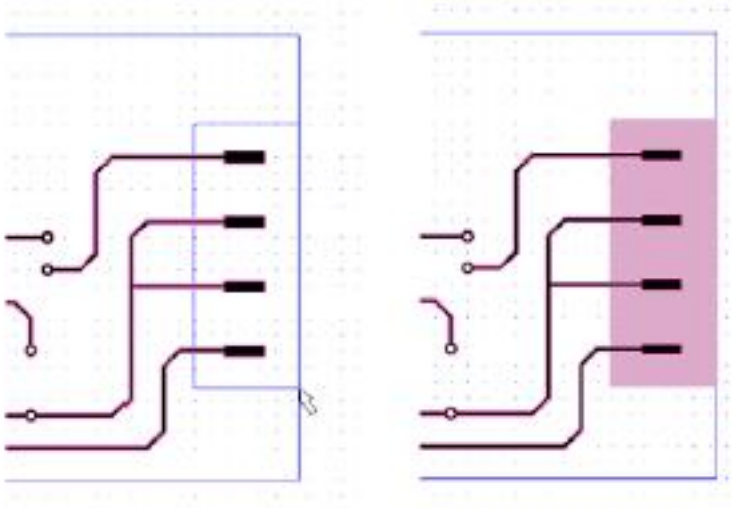
Overlap ratio:
Rubout data is overlapped to some extent so that no copper plating remains after rubout milling. The recommended value is 30%.

Minimum Length:
Set 0 usually.
Minimum Length reduces the milling lines with tiny length so that it may reduce the milling time.

Direction:
Select zigzag direction X or Y.

Area Specified using:
Select Rectangle usually.

Click Apply and select area to be rubout.



To specify the rubout area using the rectangle, first "Specify first corner" and then "Specify opposite corner" as these messages appear.

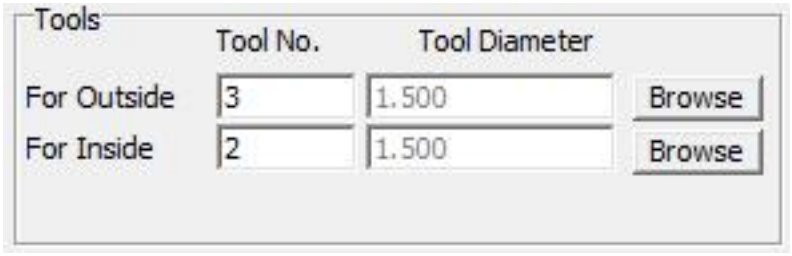
When PCB outline layer is selected as rubout area, software will search the closed drawing in the layer which is specified as layer type "PCB outline" and make rubout data in the closed drawing.

When Rubout layer is selected as rubout area, software will search the closed drawing in the layer which is specified as layer type "Rubout area" and make rubout data in the closed drawing.

Generating Contour Routing Data

This icon lets you generate contour routing data.

Layer:
Set "Layer Type" to "PCB outline". The program will search the closed figure to generate contour routing data.
The generated data will be stored in the Routing layer.



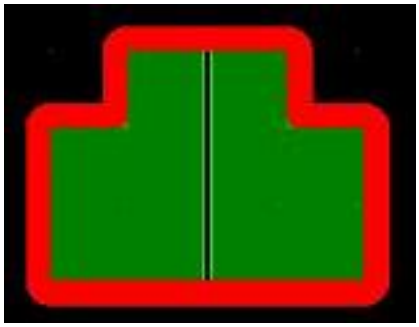
For Outside:
Click Browse to select the tool No. for routing outside of the board.

For Inside:
Select the tool No. when the data contains the area to be routing inside the board. Tool No. "For Inside" should be smaller than one "For Outside" because contour routing will process smaller tool No. first.

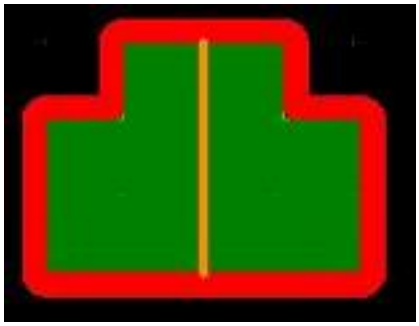
Milling Through the Gap (EASY CAD Only)

This command generates milling lines through the gap.
It is helpful for small tool bit to run the minimum path and extend the tool life.

Milling outlines cannot enter the gap which is narrower than the tool diameter.



Milling through the gap can enter this kind of gap.

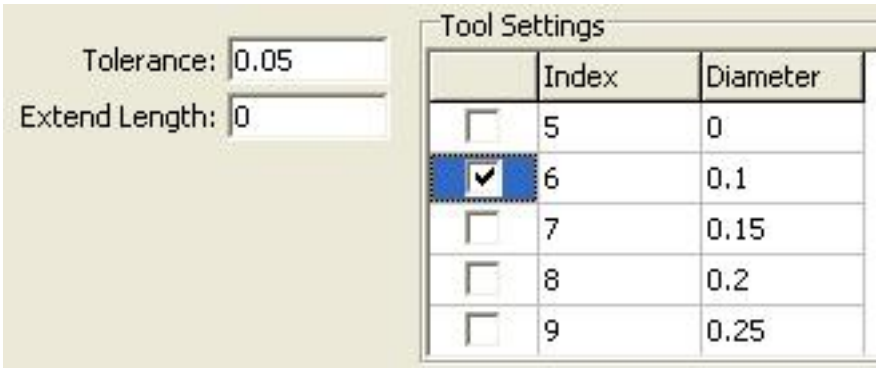


Please note that this command cannot always make the line for the all kind of gap and it depends on the layout of patterns.

Operation:

Click the icon of Milling Through the Gap

Select the tool diameter and click apply.



- Tolerance:
  This value is referred together with the selected tool diameter in caliculating the gap.
- Extended length:
  To extend milling line may prevent the remain of cut.