PCB Prototyping Machine

Auto Lab

Tutorial
MITS Electronics

REVISION:

October 1, 2011   1st edition

CONTENTS:

Design Pro Applications

There are three applications within the Design Pro software.

To change between applications, click on **Application**, located in the upper right hand corner.

EasyCAD and Converter have similar functions.

However, EasyCAD allows the user more access to manipulate the imported Gerber files. EasyCAD also allows the user to create a layout, if the user does not have access to a PCB layout software.

CAM-*** is used to control the machine for PCB fabrication.

EasyCAD	: Provides full options of CAD commands.
Converter	: Specialized for converting Gerber and DXF.
CAM-***	: Fablicates the board.

Import Gerber Files

Drag and Drop Gerber files into EasyCAD or Converter screen and import process will start.

Otherwise, select **File > Import > Gerber In**
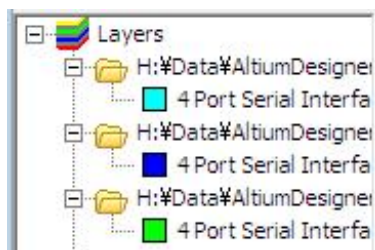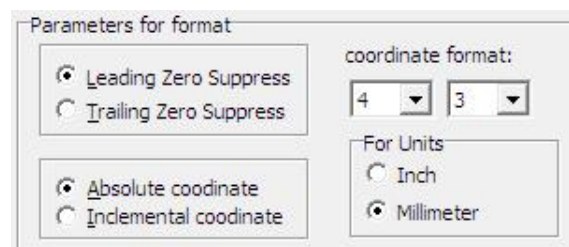Select the file and click **Open**.

Gerber Settings:
A dialog box will appear showing the basic parameters for format of the file.
When the file is RS274X format, it is not necessary to change settings.
Verify these parameters and click **Done**. This must be done for each file imported.

All of the Gerber files should appear on the workspace and the layer name added to the layer tree on the left side.

Import RS274D format:
When Gerber file is RS274D format, the default settings are displayed in this screen.
Enter the correct settings and click Done. It also needs to edit aperture list after the drawings appear on the screen

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.
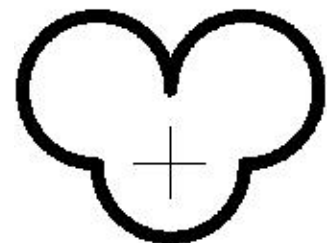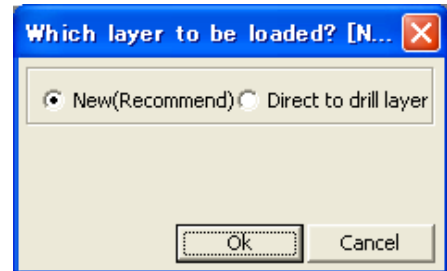
Import Drill File

Drag and Drop NC Drill file into EasyCAD or Converter screen and import process is started.
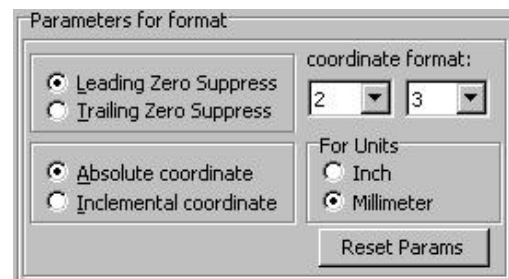
Otherwise, select **File > Import > Drill In**
Select the file and click **Open**.

A dialog box will appear asking "Which layer to be loaded?".
Click **New (Recommended)** and then click **OK**.

NC Drill Settings:
A dialog box will appear showing the basic parameters
for format of the file.

Instead of entering the settings by yourself, click
Autodetecting to detect the most suitable parameters of
drill format according to the Gerber pattern which is imported
previously.
*Please note this button doesn't appear when no
Gerber pattern is found for reference.*
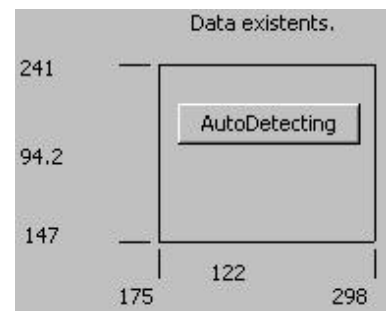
Click **Done** when the setting is finished and the hole drawings are displayed on the screen and
also the layer name is added to the layer tree on the left side.

Auto Drill

Click **Auto Drill** icon, located on the toolbar.
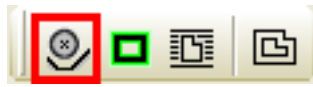
A Drill & Outline Generation Setting dialog box will appear.

Layer:
Select the **For Drill** radio button for the drill layer and **For Outline** radio button for the board outline layer.

Layer Name	for drill	for outline
H:¥Data¥AltiumDesigner_RS274X¥4 Port		
drill layer	●	○
H:¥Data¥AltiumDesigner_RS274X¥4 Port		
4 Port Serial Interface.GM1	○	●

Tolerance and Max Diameter:
Set the following parameters to:

Tolerance Drill diam. minus:	0.0300
Tolerance Drill diam. plus:	0.0300
Max. Diameter:	1.5000

a. Tolerance Drill Diam. minus: 0.030mm or 0.0015"
b. Tolerance Drill Diam. plus: 0.030mm or 0.0015"
c. Max Diameter: 1.5mm or 0.063"

Click **Apply**. All of the larger than 1.5mm (0.063") max diameter hole will appear as a board outline and be routed with the 1.5mm (0.063") contour router during fabrication.
All holes with a diameter smaller than 1.5mm (0.063") will appear as white dots and be drilled with the appropriate tools from the drill list.

How Tolerance works:
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.

Generating Milling Outlines

The milling machine cuts the outlines of pads and traces in your circuit board.
It is known as PCB isolation milling and this command creates the tool path for MITS machine.

 Click this icon and Generate Milling Outlines dialog box will appear.

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.

Tools	Tool No.	Tool Diameter	
1st time :	0	0.300	Browse
2nd time :	1	0.300	Browse
3rd time :	2	0.300	Browse

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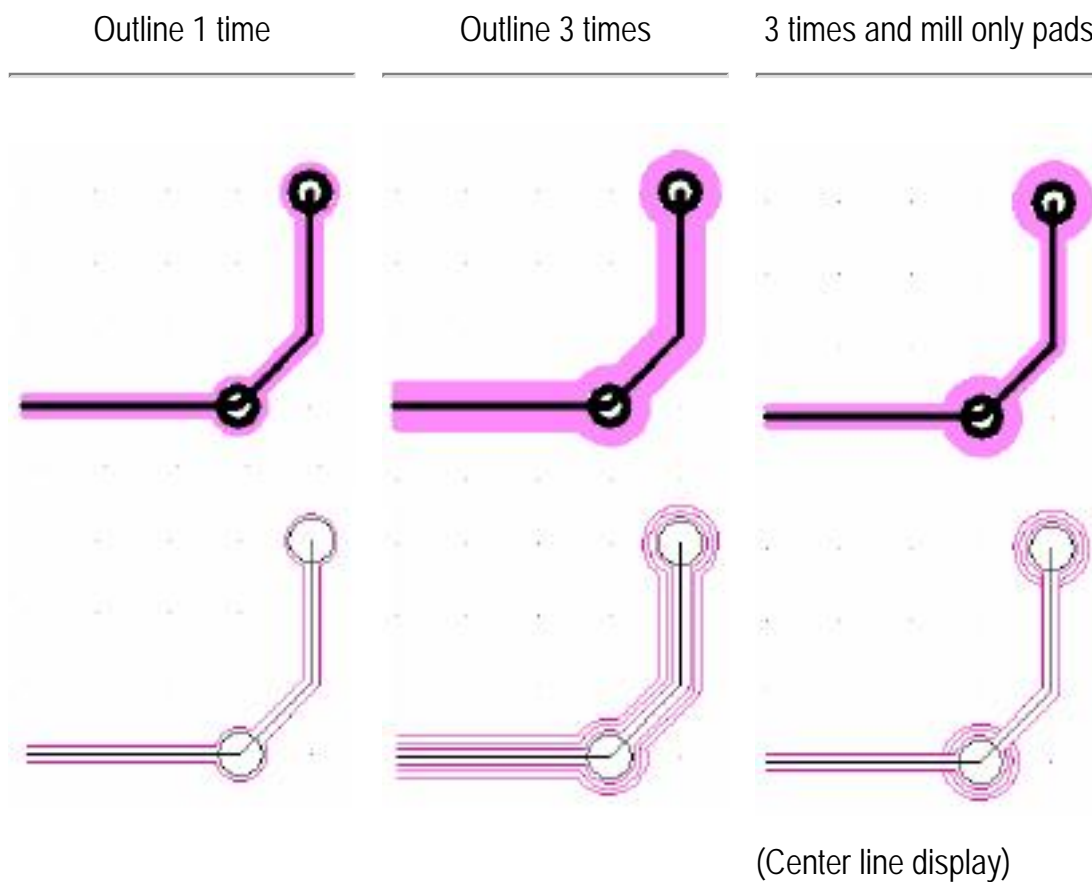
 This icon lets you change the display of width: Fill, Center Line, Ball&Stick

Generating 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
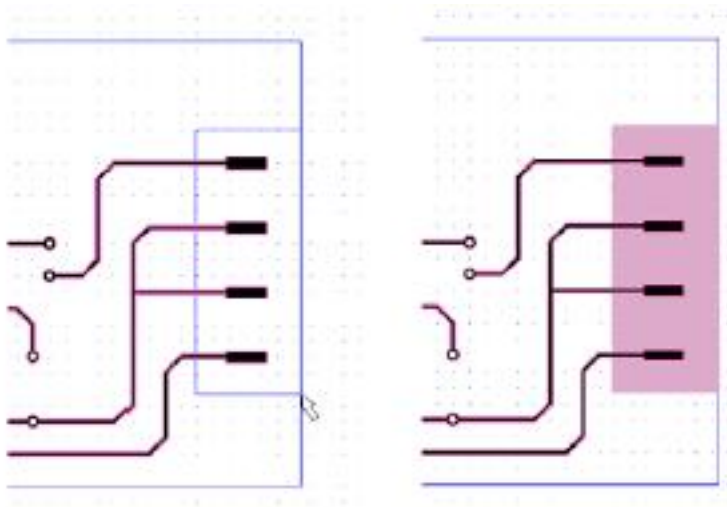the milling time.

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
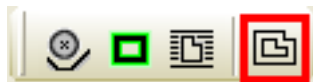
  This icon lets you generate contour routing data.

Layer Name	Visibility	Layer Type
Default		
Top	☑	Pattern Top ▾
Bottom	☑	Pattern Bottom ▾
PCBoutline	☑	PCB outline ▾

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.

Tools

	Tool No.	Tool Diameter	
For Outside	3	1.500	Browse
For Inside	2	1.500	Browse

For Outside:

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

For Inside:

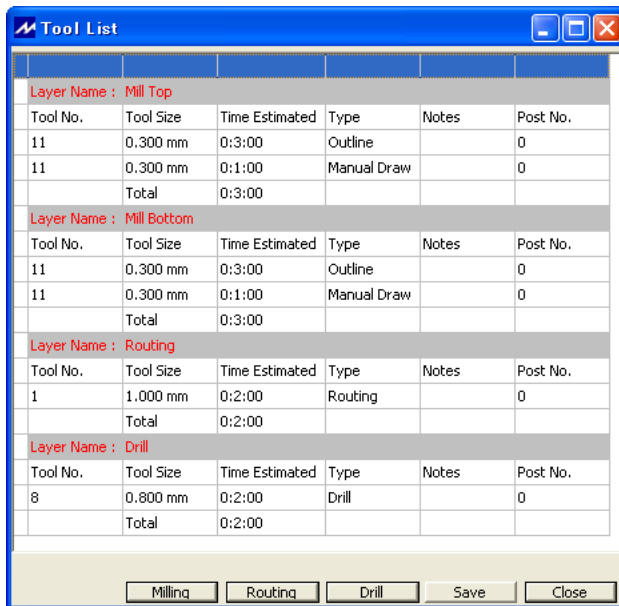Select the tool No. when the data contains the area to be routing inside the board.
Tool No. "For Inside" should be smaller than one "For Outside" because contour routing will process smaller tool No. first.

Prepare Tools for Fabricating PCB

Click **Application** and switch to CAM.
Open the file to be fabricated on the CAM screen.

Click **Information > Tool List** and a list of tools which are used in the file will appear. This list also suggests the estimated time to fabricate the file.

Tool List

Layer Name : Mill Top					
Tool No.	Tool Size	Time Estimated	Type	Notes	Post No.
11	0.300 mm	0:3:00	Outline		0
11	0.300 mm	0:1:00	Manual Draw		0
	Total	0:3:00			
Layer Name : Mill Bottom					
Tool No.	Tool Size	Time Estimated	Type	Notes	Post No.
11	0.300 mm	0:3:00	Outline		0
11	0.300 mm	0:1:00	Manual Draw		0
	Total	0:3:00			
Layer Name : Routing					
Tool No.	Tool Size	Time Estimated	Type	Notes	Post No.
1	1.000 mm	0:2:00	Routing		0
	Total	0:2:00			
Layer Name : Drill					
Tool No.	Tool Size	Time Estimated	Type	Notes	Post No.
8	0.800 mm	0:2:00	Drill		0
	Total	0:2:00			

Milling | Routing | Drill | Save | Close

Prepare the tools with proper diameter according to the list.

Milling	Drill	Routing

Set Tool to Post

Tool will be changed automatically during the machine fabricate the PCB.
You need to set the proper tool to the proper post of the machine before starting the job.
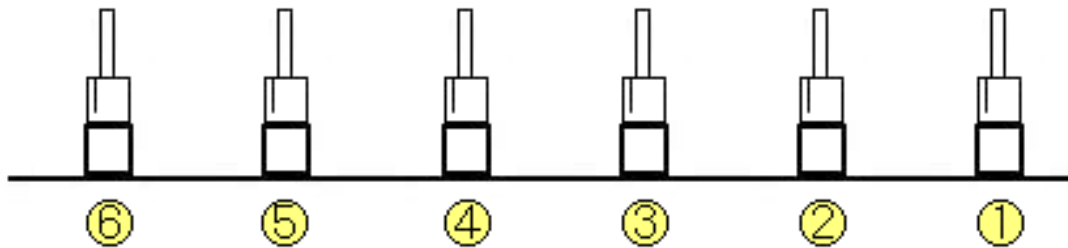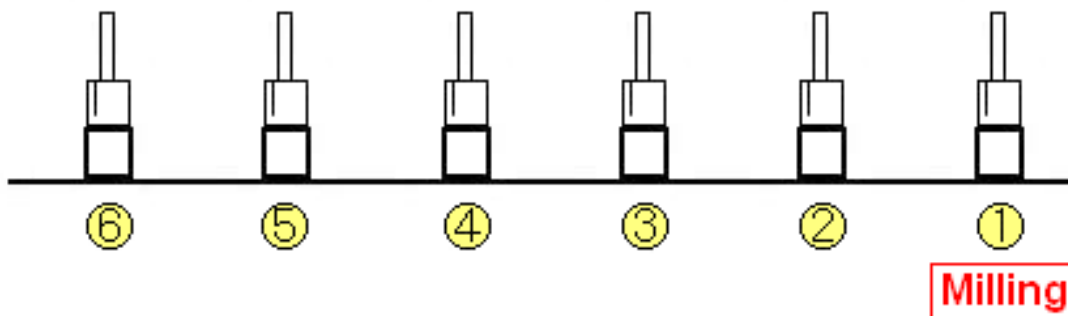

Take a look at tool list. (**Information** > **Tool List**).
In the below example, the Milling section indicates your file uses milling tool No. 0.

Layer Name :	Mill (Top)				
Tool No.	Tool Size	Time Estimated	Type	Notes	Post No.
0	0.300 mm	0:3:00	Outline		0

You may set the milling tool to the post No. 1.

Click **Milling** at the bottom of Tool List dialog box.
Milling Settings Screen will appear.

Set "1" to **Post** field in the line of
Tool No. 0.

Click **OK** to save the parameters and close dialog box.
Now the Post No. in the tool list will be also updated.

Take a look at the drill section in Tool List dialog box.
In the below example, your file uses three drills, tool No. 6, 8 and 10.

There are the setting screens for tool bits in the CAM prefs.menu:

Layer Name :	Drill				
Tool No.	Tool Size	Time Estimated	Type	Notes	Post No.
6	0.600 mm	0:1:00	Drill		0
8	0.800 mm	0:3:00	Drill		0
10	1.000 mm	0:1:00	Drill		0

You may set these drills to the post as illusted below.
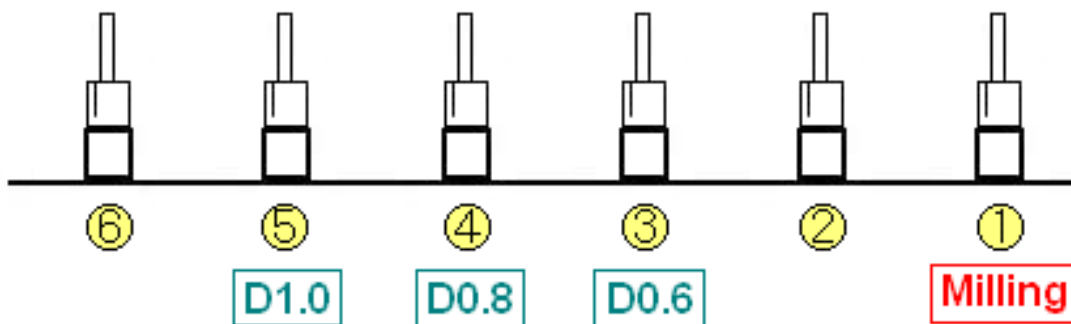
Click **Drill** at the bottom of Tool List dialog box.
Drill Setting dialog box will appear.

Set "3" to **Post** field in the lines of tool No. 6.
Set "4" to **Post** field in the line of tool No. 8.
Set "5" to **Post** field in the line of tool No. 10.
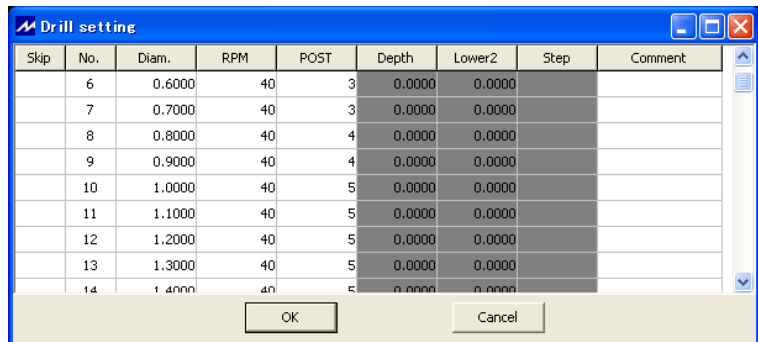
Click **OK** to close Drill Setting dialog box.
Verify the post field updated in Tool List dialog box.

Regarding routing tool settings, it is the same story.
So put the router tool to the post and edit Post in the Router Setting dialog box.

Tool Change Manually

Tool will be changed automatically during the machine fabricate the PCB. However, you will need to change tool manually in the setting up procedure.

When you see the description in this manual saying "Change the tool to milling bit" or "Install tool to the head", please do the following operations:

How to change tool manually

Click **ATC** in Manual Operation dialog box.
Auto Tool Change dialog box will appear.
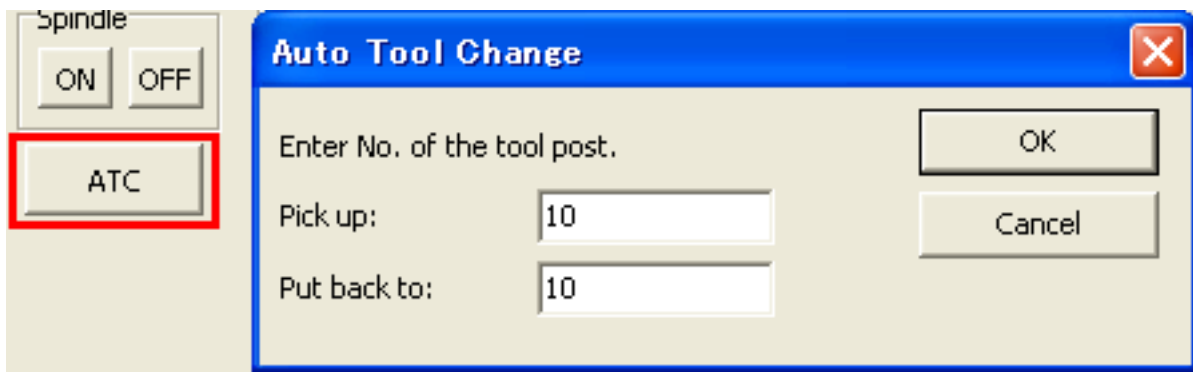
Enter Post No. from which you want to pick up in **Pick up**.

Put back to indicates the current post No. which the machine holds.
So it is not necessary to change this field in ordinary case.

Click **OK** and the head will return the tool to the post specified in **Put back to** and then pick up the tool from the post specified in **Pick up**.

Enter Thickness of the Board (Standard)

Click **CAM Prefs > Board Maker** and the screen appears.

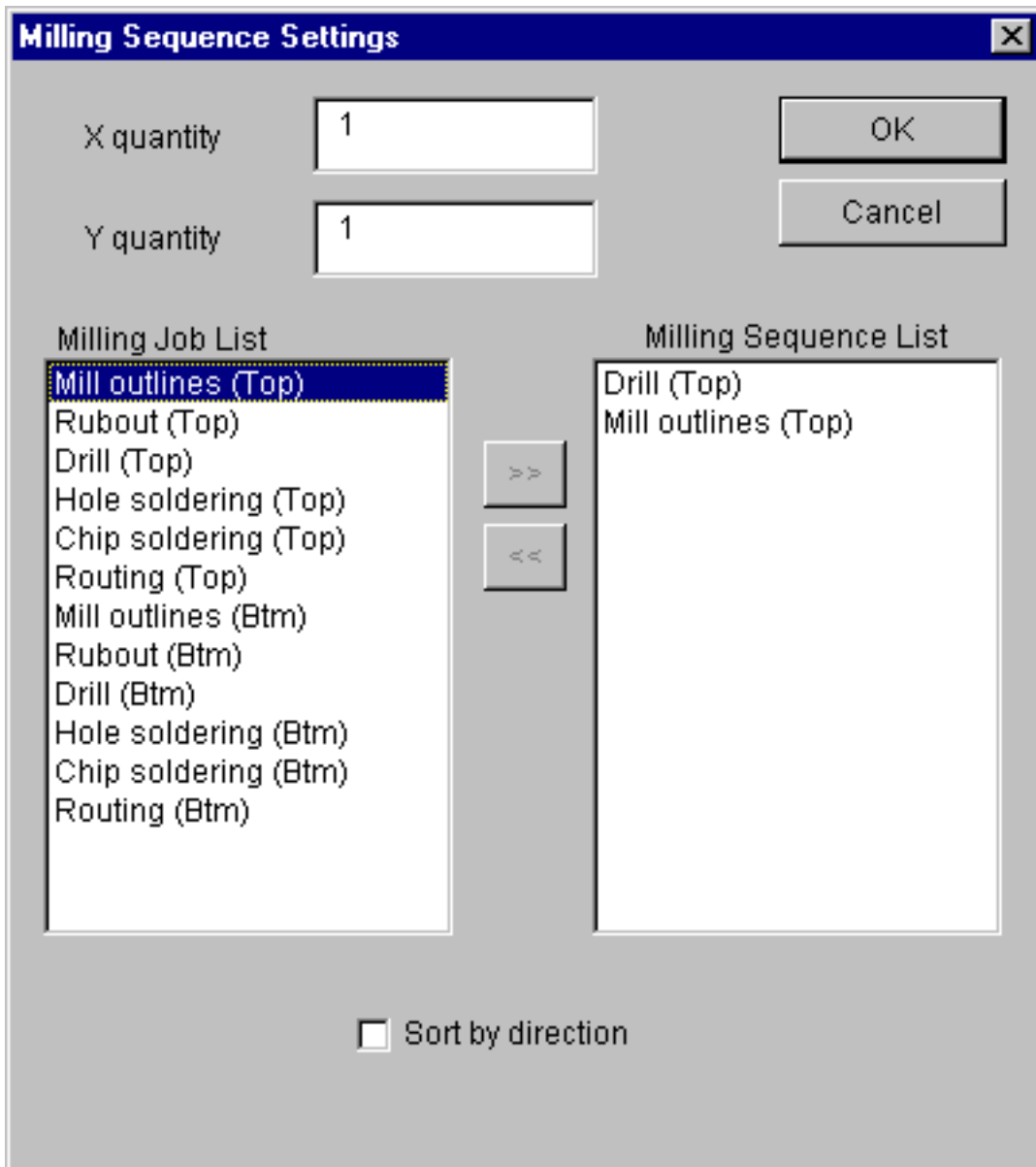| Mill spd.(mm/sec) | 12.0000 | Dry run(mm/sec) | 20.0000 |
| Lower1(mm/sec) | 12.0000 | | |
| Thickness(mm) | 1.6000 | Underlay(mm) | 1.6000 |

1. Enter Thickness of the board. (For this example, 1.6mm)

2. Enter Thickness of underlay. (For this example, 1.6mm)

Make sure that the milling machine is turned on.

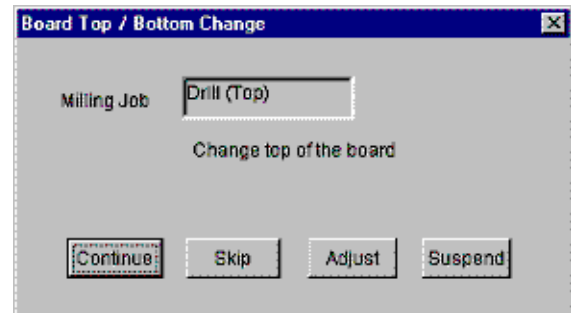Click **Start Job** icon and Milling Sequence Settings dialog box will appear.

For this example, select:

1. Drill (Top)
2. Milling outlines (Top)

Click **OK**.

Set P1 and then Confirm P2 Location

When **OK** is clicked in Milling Sequence Settings dialog box, Board Top/Bottom Change dialog box will appear.

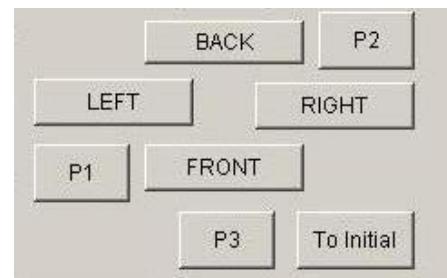Click **Adjust** to call up Manual Operation dialog box.

We need to set the milling reference point (P1).

Click the move buttons (FRONT, BACK, etc.) to move the head to any location on the board.
Click **Set P1-P2**, and that location will become P1.
And a dialog box will appear asking "Align P2?"
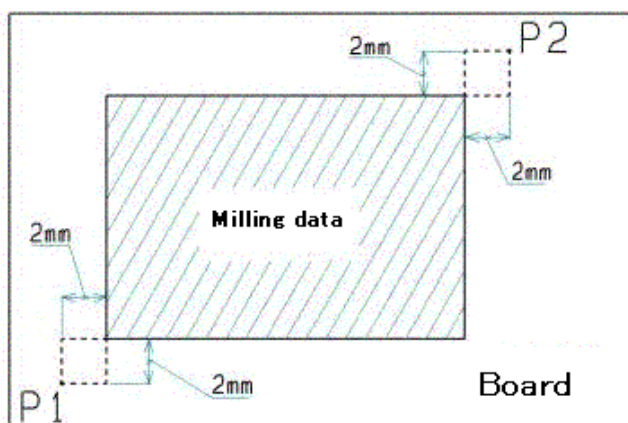Click **No** for now.

When P1 has been set, we need to check the location of P2:
Click **P2** and the head will move to P2.
Make sure that the milling area fits within the board.
If you want to change P1, move the head again to the desired location and click **Set P1-P2**.

It is very important to check the location of P2. If the head goes outside of the board during milling, it may cause accident such as breaking the bit, hurt the surface of the table and so on.

Set Distance from Spindle to Camera

The camera is mounted beside the head of MITS machine.
It is necessary to set the distance between the spindle and the camera.

When you have set P1 location on Manual Operation dialog box,
we recommend to adjust the distance of camera in the following procedures.

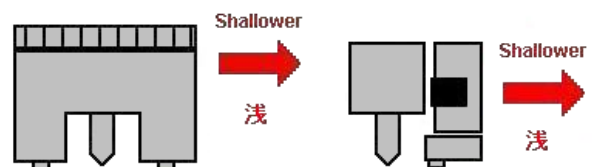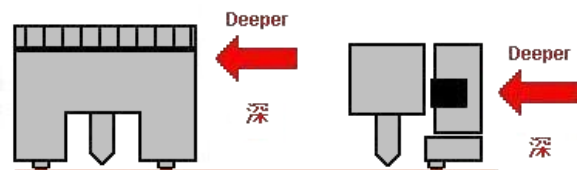Install the milling tool to the head.

☑ **Milling**　　　　Drill　　　　　　Routing

If the tip of tool sticks out of the bottom of
pressure foot too much, turn the depth
screw to shallower.

Click **ON** to rotate the spindle motor.
Click **DOWN** to lower the head.
If the tip of tool doesn't touch the surface of
the board, turn the depth screw to deeper.

Confirm the tip of tool touch the board surface.
Click **UP** to raise the head.
Click **OFF** to stop the spindle.
Now you will see the milling mark on the P1 point.
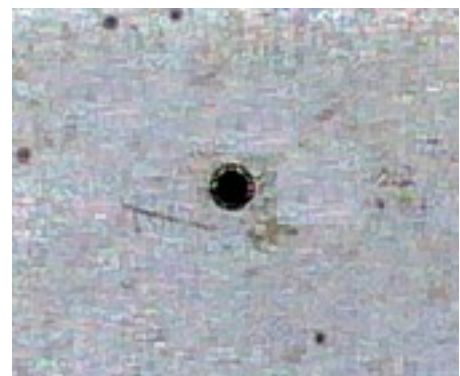
Click **CAMERA**.
Start Design View software if it is not started.
Now you are ready to see the camera view.

Click **Offset** located below **Camera**
button
and a dialog box will appear saying "Align
center of camera with reference of spindle".
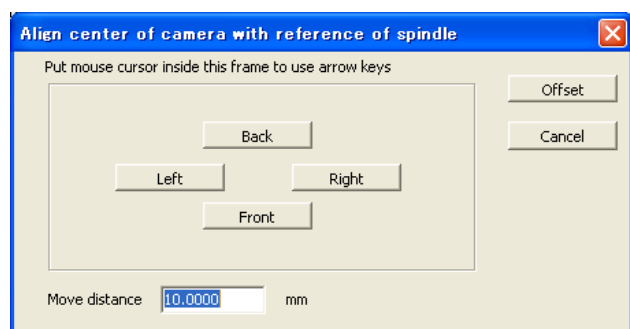
Move the camera to the location where the marker display matches the milling mark at P1.

Then click **Offset** on the dialog box.

The program will caliculate the distance from spindle motor to camera.

Click Marker button on Design View.
Marker Prefernce screen enables you to change marker shape and color.
It helps you to see the marker easier.
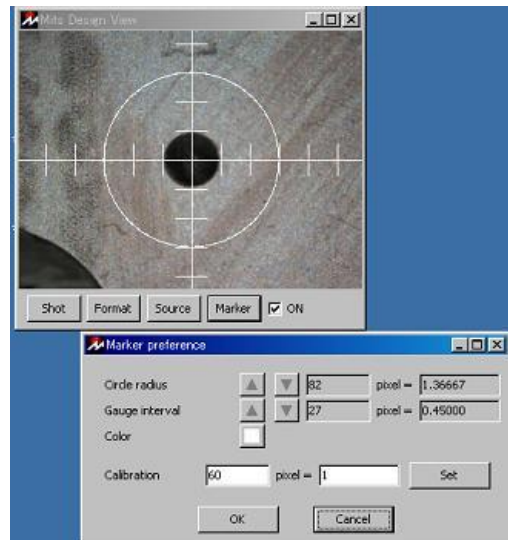
Adjust Depth of Cut using Camera

Let's make a test cut to check the width of the channel cut by the milling tool.

Change the tool to the milling bit if it is not installed.
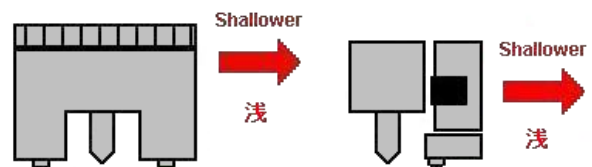
☑ **Milling**   Drill   Routing

If the tip of tool sticks out of the bottom of
pressure foot too much, turn the depth
screw to shallower.

From now, we will make a trial cut.
Move the head to the outside of P1-P2 milling area to prevent from hurting the surface inside the
P1-P2 area. (use the FRONT, BACK, LEFT, RIGHT move buttons as needed)

Click **CAMERA** to see the camera view.
Click **ON** to rotate the spindle motor.
Click **DOWN** to lower the head.

The head moves with camera offset distance and goes down.
Turn the depth adjustment screw deeper or shallower
so that the milling tool touches on the board surface.

In this position, press one of the move buttons once and the head will move.
Now you have obtained a trial cut.

Click **UP** to raise the head.
The head goes up and move with camera offset distance.
You will see the trial cut in Design View screen.

Click **OFF** to stop the spindle motor.

Check the width of the channel that was cut.

Adjust the milling depth and make a trial cut several
times so that the width of this channel is the same as
the diameter of the tool specified when milling
outlines is generated in EasyCAD or Converter.

When the adjustment is finshed, click **CAMERA** to
turn off.
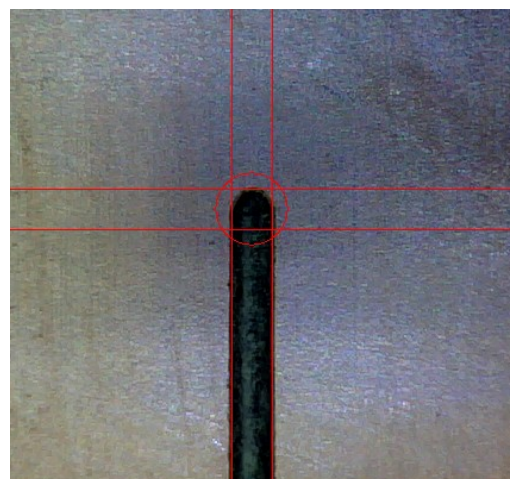
Drill P1 and P2 Location

After setting the reference point, we need to make the reference holes for P1 and P2.

Reference holes are used for the alignment using camera system.
Also, making these holes are useful for understanding how large the milling area is.

Change the tool to drill.
For example, 0.8mm (0.032") drill.

Milling   ✓ **Drill**   Routing

Click **Drill P1-P2** and the head will move the P2 point.
A dialog box will appear asking "Ready to drill P1 and P2?".

Click **Yes** and the head will start to drill holes on P2 and P1.

When everything is done, click **OK** to close Manual Operation dialog box.
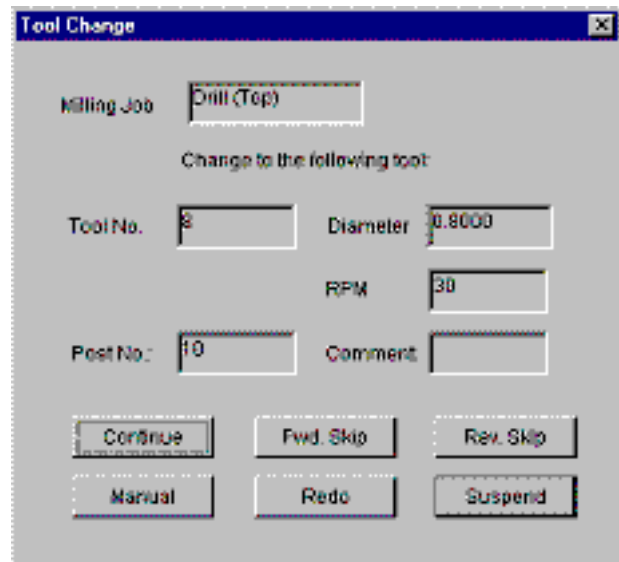It goes back to Board Top/Bottom Change dialog box.

How to Continue Drill and Milling

Board Top / Bottom Change

Milling Job: Drill (Top)

Change top of the board

[Continue] [Skip] [Adjust] [Suspend]
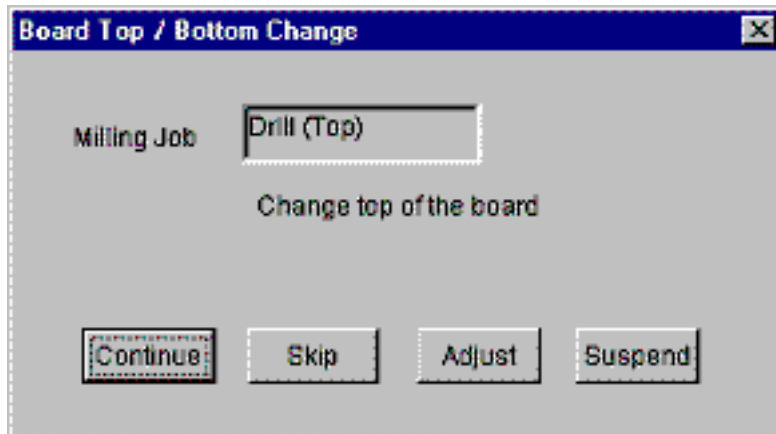
Click **Continue** in Board Top/ Bottom dialog box.

A tool change dialog box will appear when the milling machine needs the tool change manually.
Install the proper tool displayed in the dialog box.

Click **Continue** so that MITS machine will start the drill and milling job.

Tool Change

Milling Job: Drill (Top)

Change to the following tool:

Tool No.: 8   Diameter: 0.8000

RPM: 30

Post No.: 10   Comment:

[Continue] [Fwd. Skip] [Rev. Skip]
[Manual] [Redo] [Suspend]

DO NOT touch the spindle motor or allow your face to get near it during milling.

To interrupt the milling operation, press ESC key. After finishing the milling data which has been already sent, the operation will stop.
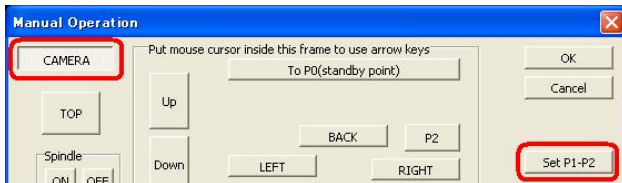
If emergency stop is needed, turn off the power switch.

Alignment for Double Sided using Camera

Make sure that P1 and P2 location are drilled before flipping over the board.
If not, refer to the instructions of "Drill P1/P2".

In Board Top/Bottom Change dialog box, click **Adjust** to call up Manual Operation dialog box.

Click **CAMERA** to see the camera view.

Move the camera right on the P1 hole, click **set P1-P2**.

A dialog box will appear asking "Align P2?".
Click **Yes** to align P2.
The camera will move to the location near the P2 hole.

Move the camera right on the P2 hole.
Click **Set P1-P2** again.
Software calculates any deviation in board alignment and compensates as needed.

Click **CAMERA** to turn off.
Click **OK** to close Manual Operation dialog box.