Eagle Excellon to G-code converter

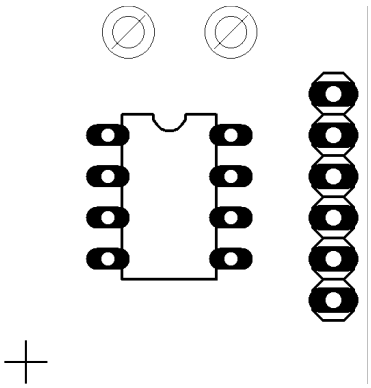
Lindsay R. Wilson lindsay@imajeenyus.com
Program written 08/07/10, instructions written 23/04/11

Summary

Although Eagle can directly output a file containing drilling information, it is not in a format suitable for direct use by a CNC machine. This program provides a means of generating a G-code file from an Eagle board file, which can then be used on a CNC machine for automatic hole drilling. It can also perform full scaling and skewing correction to correct for errors in home-made PCB masks and boards.

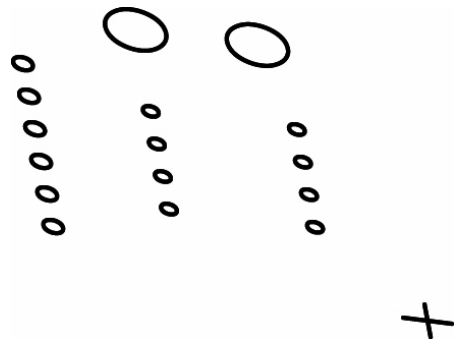
Example of use

The best way to demonstrate how to use the program is with an example. We will use the board shown below ("example.brd"). This view is from the TOP and is what's seen from within Eagle itself.



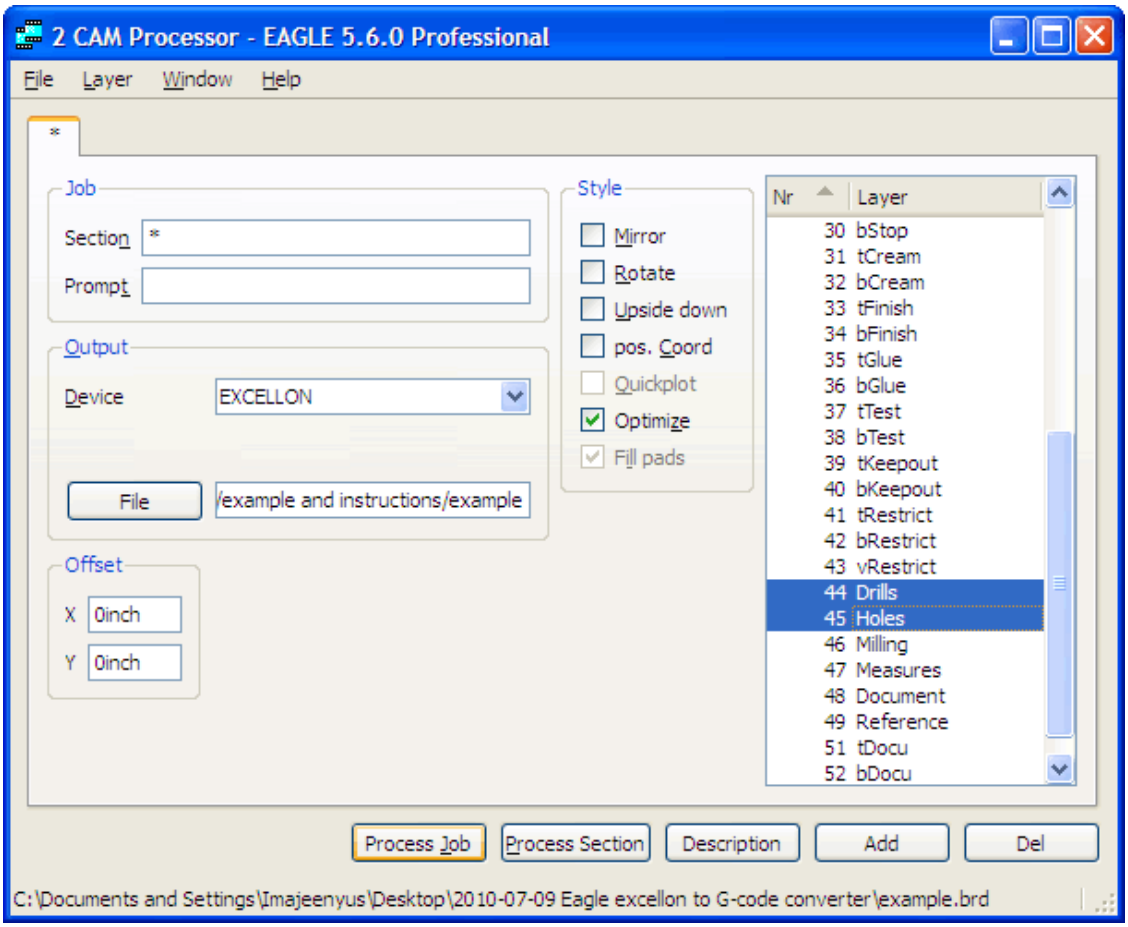
Three different drill sizes are used in this board. The DIP8 IC uses 0.032" holes, the 6-way pin header uses 0.040" holes, and the two large holes at the top are 0.125". The origin is at the bottom-left.

Now, let's suppose that our laser printer is extremely wonky and we end up with the board shown below. This is viewed from the BOTTOM; the origin is therefore at the bottom-right, shown by the cross.



The board is clearly very badly skewed and scaled in both X and Y axes. However, this can be corrected using the program. To do this, the PCB is mounted on the bed of the CNC drilling machine, and a video camera mounted on the head is used to digitise the coordinates of several of the holes on the PCB. These values are fed into the conversion program and it corrects for any distortion. The program assumes that the board is placed on the drilling machine BOTTOM-UP, i.e. it is drilled FROM THE BOTTOM.

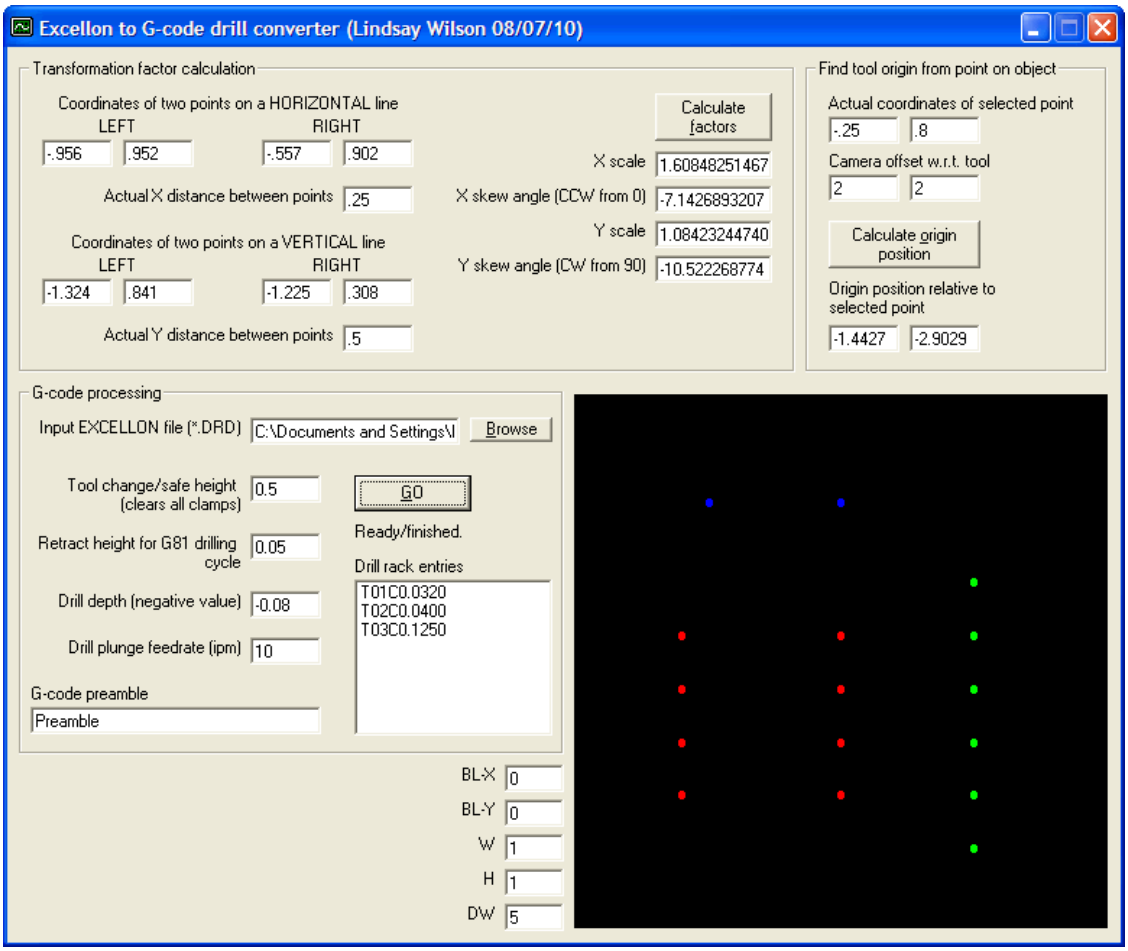
First, we must export drill hole information from Eagle. Open the board and go to the CAM processor. Choose Excellon as the output device. Untick all boxes except Optimise. Deselect all layers except Drills and Holes. Make sure the Offset values are 0 for both X and Y. Enter a filename with a *.drd extension to save the data as, and click Process Job. The screencap below shows the CAM processor dialog.



This produces two files; a *.drd and a *.dri. Ignore the *.dri. The text overleaf shows the contents of the *.drd file. As you can see, three different drills are used, identified as T01, T02 and T03, and the X,Y coordinates of each hole are listed, expressed in thou.

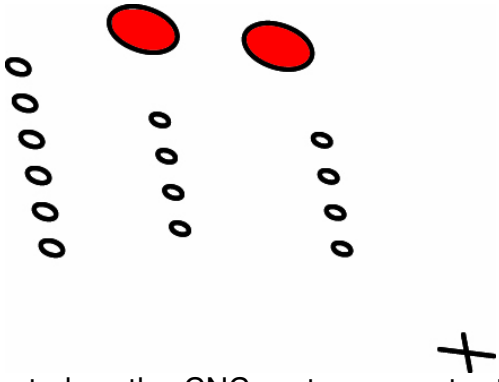
```
%
M48
M72
T01C0.0320
T02C0.0400
T03C0.1250
%
T01
X2000Y2500
X2000Y3500
X2000Y4500
X2000Y5500
X5000Y5500
X5000Y4500
X5000Y3500
X5000Y2500
T02
X7500Y2500
X7500Y1500
X7500Y3500
X7500Y4500
X7500Y5500
X7500Y6500
T03
X5000Y8000
X2500Y8000
M30
```

Open up the conversion program. A screenshot is shown below – refer to this.



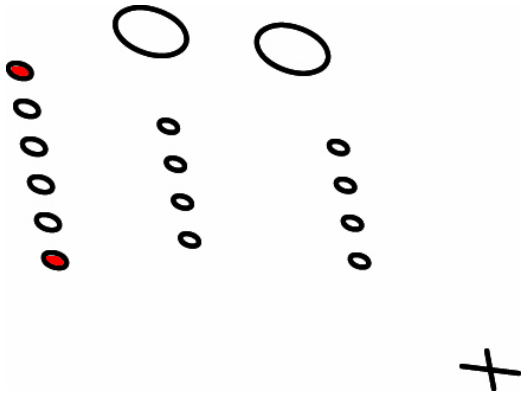
Transformation factor calculation. This allows the program to correct for distortion and misalignment of the PCB board. Note that the CNC machine does not need to be zeroed on the origin for this – we will zero it later.

First, we need the coordinates of two points which are SUPPOSED to be on a horizontal line. Here, we will use the two 0.125" holes, indicated in red below.



Using the camera mounted on the CNC router, we enter the coordinates (in the real world) of these two holes, remembering to enter the left and right holes properly. We also enter the actual horizontal spacing between these holes, which is 0.25". The program now automatically calculates the X scale and skew angles.

Next, we repeat this for the Y axis. This time we will use the top and bottom holes of the 6-way header, indicated in red below.

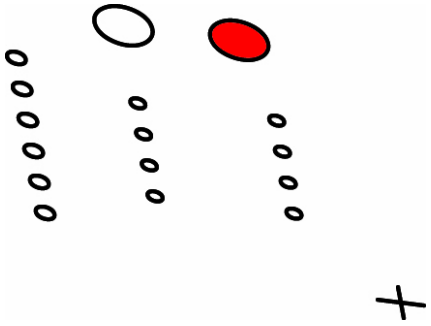


Again, we enter the coordinates of these two holes, and the actual vertical spacing between them, 0.5".

Having determined the transformation factors, we can now zero the machine.

Find tool origin from point on object

This tells us the relative distance to move so as to align the actual tool axis with the origin of the PCB board. We use the camera to centre on a particular hole, for example the right-hand 0.125" hole, shown in red overleaf.



The actual coordinates of this hole should be -0.25,0.8 and we enter this in the appropriate boxes. We will also have determined previously the camera offset relative to the tool, and we also enter this (2,2 is used as an example). “Origin position relative to selected point” are the distances we have to move to bring the tool into line with the origin of the PCB board, where we then zero the machine.

Now we can process the G-code.

G-code processing

Choose the input Excellon drill file. Tool change/safe height is the height which is above all the clamps used to affix the PCB to the bed of the machine. Retract height is the height above the PCB surface to which the drill moves between holes. This is much less than the tool change/safe height, because it speeds drilling. Drill depth is the depth below the PCB surface to which the hole is drilled. Drill plunge federate is just what it says.

If desired, you can enter a comment into G-code preamble.

Click Go. The different drill rack entries should appear in the table, and a drawing showing the various holes (as seen from the board TOP) should appear.

The program produces one *.ngc G-code file for each drill size used. Here are the contents of T01C0.0320.ngc:

```
(Preamble)
(Toolrack: T01C0.0320)
G00 Z .5
G81 X-.3687 Y .3065 Z-.08 R .05 F 10
G81 X-.3885 Y .4131 Z-.08 R .05 F 10
G81 X-.4083 Y .5197 Z-.08 R .05 F 10
G81 X-.4281 Y .6263 Z-.08 R .05 F 10
G81 X-.9069 Y .6863 Z-.08 R .05 F 10
G81 X-.8871 Y .5797 Z-.08 R .05 F 10
G81 X-.8673 Y .4731 Z-.08 R .05 F 10
G81 X-.8475 Y .3665 Z-.08 R .05 F 10
G80
G00 Z .5
G00 X0 Y0
M02
```

As you can see, it contains all the required G-code commands to drill the board.

Drawing

The holes are shown in the area at the bottom-right of the window. BL-X, BL-Y are the bottom-left coordinates of the window. W, H are the width and height. DW is the drawwidth in pixels of the holes. All dimensions are in inches.

Seven different colours are available, corresponding to the different drill sizes. If more than seven different drill sizes are used, other ones all appear as white.

What if my CNC machine is metric?

Not to worry! For the transformation factor calculation, enter the metric point coordinates, but enter the imperial distances between the points. This will result in a suitable scaling factor. Also, just enter suitable metric values into the G-code processing boxes.

Help!

Questions, comments, please email me at lindsay@imajeenyus.com