

# PCB Production Methods

# Outline

- Motivation
- File Formats
  - Excellon Drill Files
  - Gerber
- A Typical PCB Manufacturing Process
- Populating and Soldering
- Hobbyist Solutions
- Conclusion

# Motivation

- Understanding underlying manufacturing processes is almost always important for an engineer:
  - Allows the design to exploit capabilities; and
  - Ensures that the design will be manufacturable.
- PCBs have been produced for many years; advances with technological improvements are notable.

# File Formats

- As discussed in a previous lecture, professional manufacture of PCBs is typically done through the use of CAD-independent files:
  - Gerber Files – describe the copper foil layout
  - Drill Files (often in the Excellon format) – describe the location and size of holes
- Both are meant for direct use with automated PCB production equipment.

# File Formats: Excellon Drill File

- Excellon Files provide a command sequence to a system that drills PCBs
- This equipment (or its computer driver) has an interpreter that receives and executes the commands in sequence.
- The next two slides describe the list of commands that are interpreted.

Source: “Norme Excellon”, available at [http://www.forelec.ch/www/norme\\_excellon.htm](http://www.forelec.ch/www/norme_excellon.htm), accessed 12 Feb 2008 (dead link at 26 Jan 2011).

Excellon file format is governed by ANSI/IPC-NC-349 .

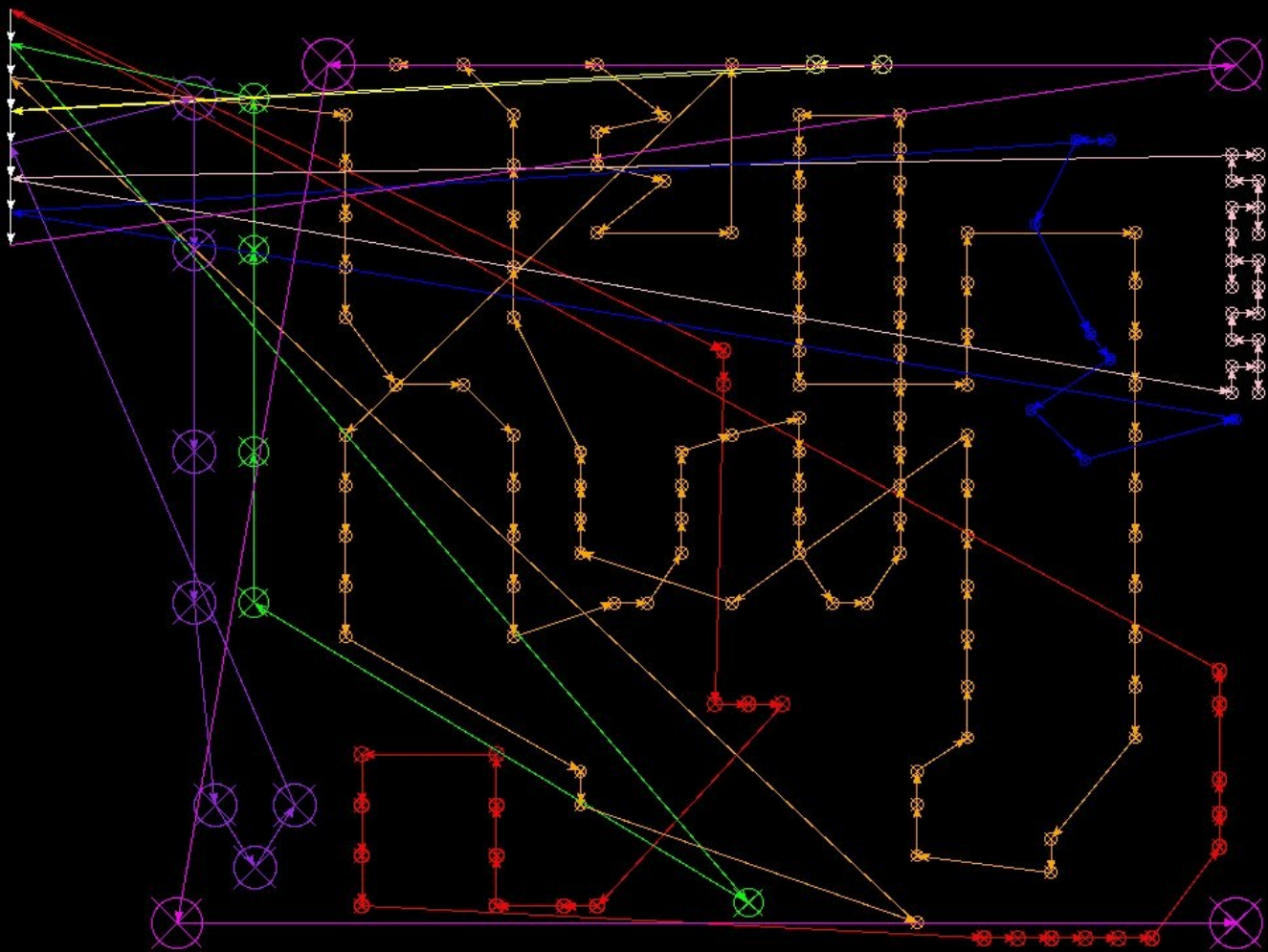
%	Rewind and Stop
X#Y#	Move and Drill
T#	Tool Selection
M30	End of Program
M00	End of Program
M25	Beginning of Pattern
M31	Beginning of Pattern
M01	End of Pattern
M02 X#Y#	Repeat Pattern
R#M02X#Y#	Multiple Repeat Pattern
M02 X#Y# M70	Swap Axis
M02 X#Y# M80	Mirror Image X Axis
M02 X#Y# M90	Mirror Image Y Axis
M08	End of Step and Repeat
N#	Block Sequence Number
/	Block Delete

R#X#Y#	Repeat Hole
G05, G81	Select Drill Mode
G04 X#	Variable Dwell (ignored)
G90	Absolute Mode
G91	Incremental Mode
G92 X#Y#	Set Zero
G93 X#Y#	Set Zero
M48	Program Header to first "%"
M47	Operator Message CRT Display
M71	Metric Mode
M72	English-Imperial Mode
Snn	Spindle Speed (RPM)
Fnn	Z axis feed speed (IPM)

Typically, only a subset of these commands are used.

# File Formats: Excellon Drill File Example

%	Reset and rewind.
M48	Start of header.
M72	Imperial (English) Mode: units in inches
T01C0.0420	Tool 1 Change: to 42 mil
T02C0.0860	Tool 2 Change: to 86 mil
T03C0.0350	:
T04C0.0520	:
T05C0.1250	:
T06C0.0354	:
T07C0.0280	:
T08C0.1520	Tool 8 Change: 152 mil
%	End of Header: Drill data follows
T01	Select Tool 1 (42 mil)
X2120Y1112	Drill at (2120 mil,1112 mil)
: (Lots of data removed)	
T08	Select Tool 8 (152 mil)
X3645Y262	Drill at (3645 mil, 262 mil)
: (Data removed)	
M30	End of program.



# File Formats: Excellon Drill File

- Sample of a drilling machine doing its job:

<http://www.youtube.com/watch?v=7Fda-rVhqFg>

# File Formats: Gerber

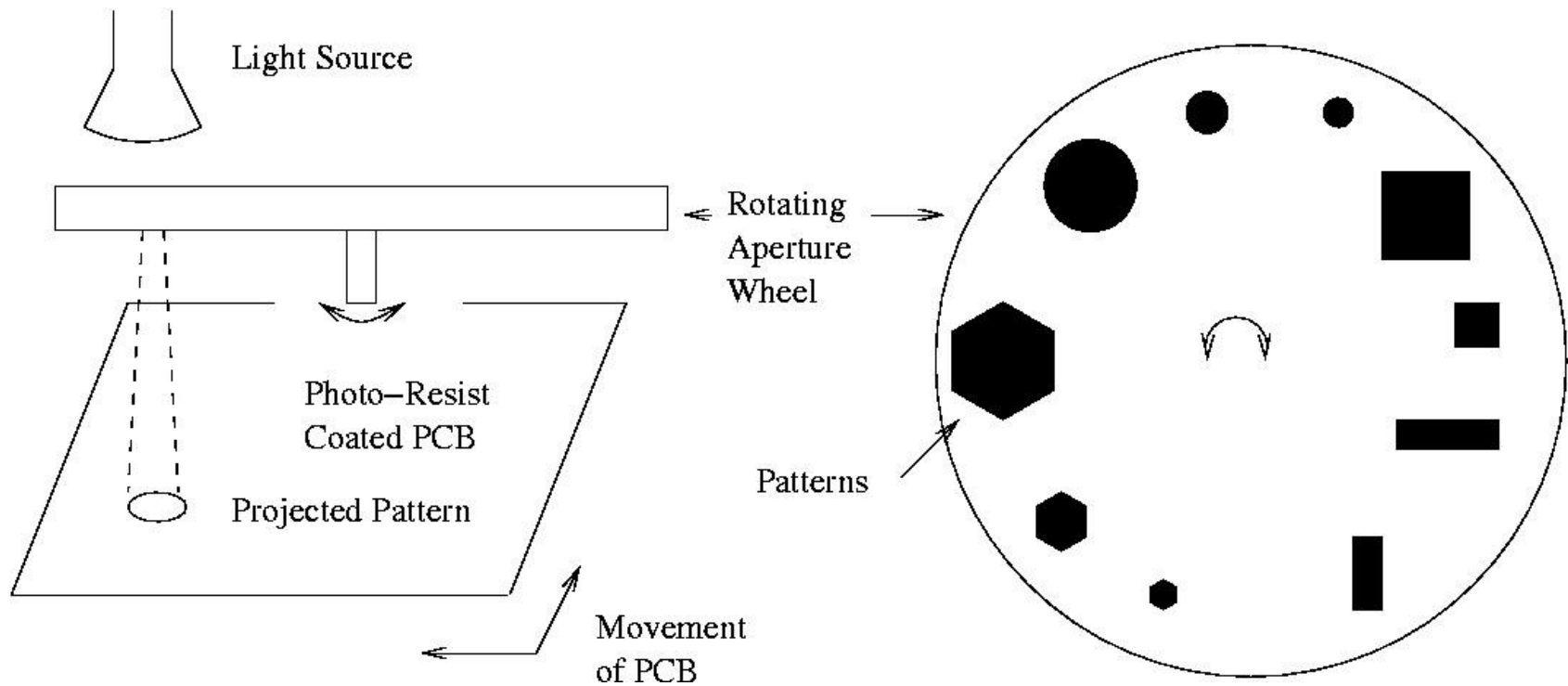
- Gerber files are typically used to describe the copper foil patterns on the PCB
- The “interpreter” idea is similar to what was described for the Excellon format
- Since pad shapes and track sizes need to be specified, many new commands are used
- These commands are not covered here, but a good source of information can be found in “Gerber RS-274X Format User’s Guide”, Barco Graphics, N.V., Gent, Belgium, 1998.

[http://www.ucamco.com/public/RS-274X\\_Extended\\_Gerber\\_Format\\_Specification\\_201201.pdf](http://www.ucamco.com/public/RS-274X_Extended_Gerber_Format_Specification_201201.pdf)

# File Formats: Gerber

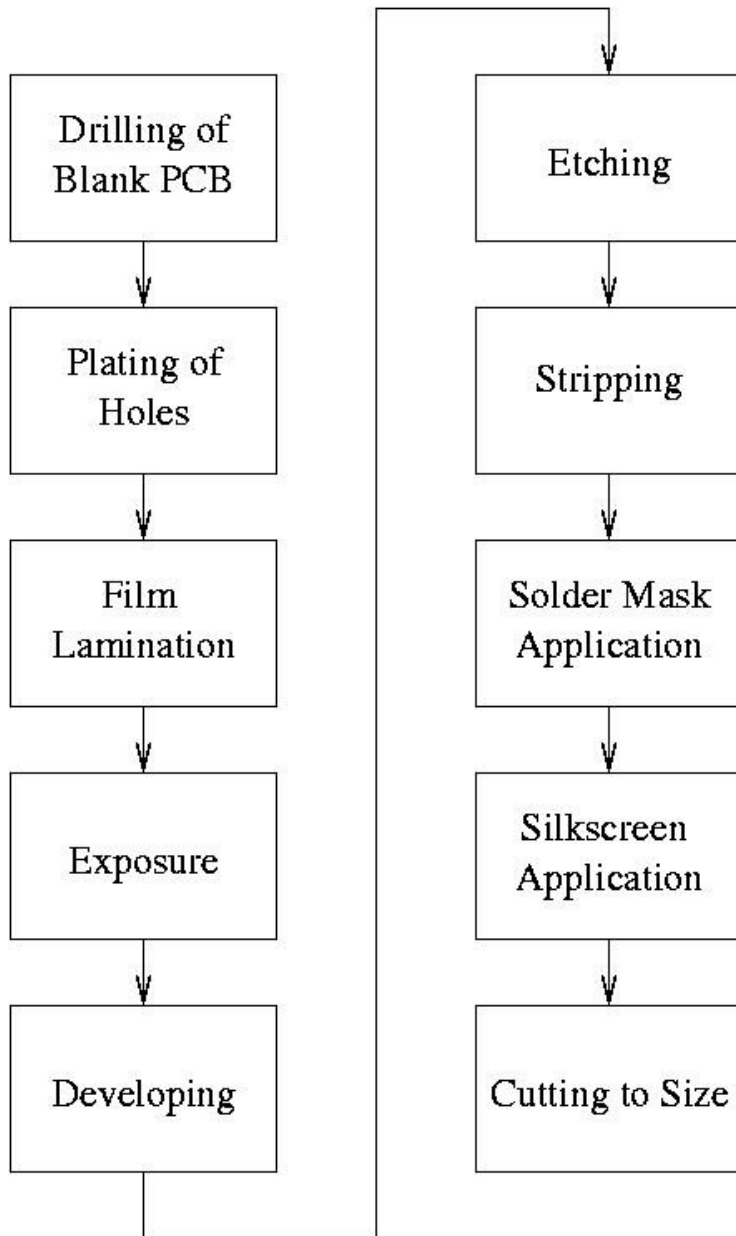
- Much of the terminology originates from the manner in which PCBs are often made: through exposure of a photo-resist layer on a board:
  - Aperture: a size and shape of a pad (or, if moved while “on”, the width of a trace and the shape of its ends).
  - Aperture Wheel: the set of apertures available. This is analogous to the set of drills available in the Excellon “Drill Rack”.

# File Formats: Gerber – Dated Photoplotter



# Typical PCB Manufacturing Process

- We have already examined some of the tolerances of modern production methods.
- In this section, we will look at some of the equipment that is used in modern PCB production.



Each development stage is carried out by a dedicated piece of equipment – which may form part of an assembly line.

Depending on your application, note that some stages may be missing here. For instance, a tinning stage is often included after etching to reduce copper layer oxidation. A multi-layer PCB, too, requires a lamination stage to join the layers.

\*Cleaning is required between most stages.



A multi-stage plate-through hole setup from Mega Electronics



A raster plotter from Mega Electronics serves as a replacement to the photoplotter described earlier. This plotter is capable of creating a 5 micron spot.



A laminator is used to apply the photoresist layer to the PCB. This laminator is from Mega Electronics.

Source: <http://www.megauk.com> accessed 12 Feb 2008



UV units used to expose the photoresist on a PCB.



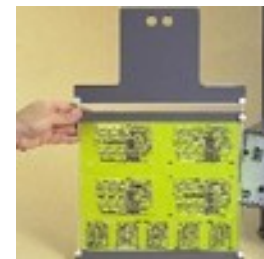
There are many different sorts of etching, developing, and stripping setups. This one is from Mega Electronics and is a unit that sprays the chemicals over board panels, mounted inside.

Different units (of the same type) are typically used for each stage described, above.

<http://www.youtube.com/watch?v=e78q633ObxE>

Shows a board being etched.

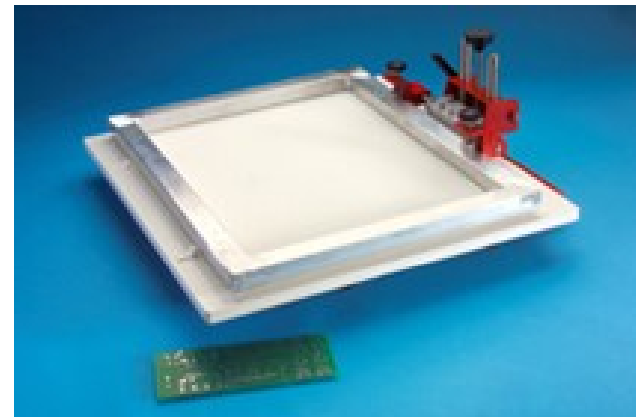
A panel that is inserted into a spray tank.





Again from Mega Electronics, units such as this “Production Line” can be used to combine the chemical stages into a convenient unit. The operator is responsible for forwarding the PCB through the stages.

Etch-resist, solder masks, parts placement silkscreen legends, and even solder paste can be applied to a PCB using silkscreen printing technology.



# Populating and Soldering

- Once a board is created, it is still necessary to “stuff” the board with components.
- This can be done by hand, but in mass-production time is an issue.
- There are two typical ways that a board has components soldered onto it:
  - The Reflow method where solder is applied to the board in a paste form, the components are added and then soldering is performed in an oven; and
  - The Wave soldering method sees the components added, and then passed over a “standing wave” of solder to perform the soldering.

# Populating and Soldering

- Each of the two methods has advantages and disadvantages:
  - Some parts cannot handle the thermal shock experienced by being passed through molten solder;
  - When wave soldering is used, it is necessary to hold components in place: glue!
  - Wave soldering is FAST!
  - Reflow soldering does not really work with through-hole parts.
- Regardless of the method used, particular temperature profiles must be adhered to.

# Populating and Soldering: Solder Paste Application for Reflow Soldering

- “Solder Paste” is solder in a very fine form suspended in flux.
- It can be placed on a PCB via a silkscreen, or using a needle applicator:  
CNC Taig Mill Solder Paste Dispenser  
<http://www.youtube.com/watch?v=HdqVt0jCBHk>
- The paste locations are dictated by the corresponding Gerber output from the CAD package.

# Populating and Soldering: Pick and Place Units

- To place components on a PCB, a “Pick and Place” unit can be used.
- Again, component placement information originates from the CAD package.
- An interesting home-brewed solution that shows many of the necessary steps in this process is at:  
Homebrew Surface Mount Pick and Place Taig Mill Conversion  
[http://www.youtube.com/watch?v=\\_\\_dEMKzkLYc](http://www.youtube.com/watch?v=__dEMKzkLYc)
- A commercial unit in operation can be seen at:  
<http://www.youtube.com/watch?v=JvexfPCFhgo>

<http://www.youtube.com/watch?v=sw0XPI25hc8> shows a solder standing wave nicely.



# Hobbyist Solutions

- It is possible to make pretty decent PCBs without spending much money.
- Follow all safety protocols!
- Do not flush chemicals down the drain!  
Save used chemicals in a clearly labelled container and turn them in to the EcoStation.
- If on campus, there is a chemical handling protocol that needs to be followed.

# Hobbyist Solutions

- As with the professional production methods, there are many possibilities of how even a hobbyist can manufacture PCBs.
- A “toner transfer” method that gives reasonable results is described at:

<http://www.dr-lex.be/hardware/tonertransfer.html>

- A video of someone going through this process can be found at:

[http://www.youtube.com/watch?v=tTxPnZLpp\\_8](http://www.youtube.com/watch?v=tTxPnZLpp_8)

Variations: using a printer fuser instead of an iron, using a powder-based etchant, and tinning the board using solder paste and a heat gun.

**Please handle chemicals in a safer manner!**

# Conclusion

- You now have an idea of what the PCB production process involves
  - At a professional level; and
  - At a hobbyist level.
- Hopefully with this knowledge you have gained enough insight to allow the design of PCBs that are compatible with the means by which they are created.