What is “surface mount”?

A way of attaching electronic components to a printed circuit board

The solder joint forms the mechanical and electrical connection

Bonding of the solder joint is to the surface of a conductive land pattern

Connection does not use through holes or terminals
Surface Mount vs. Through Hole
Advantages of SMT

• smaller parts
• denser layout
• cheaper pcbs (no holes to drill)
• improved shock and vibration characteristics
• improved frequency response
• easier to shield from EMI / RFI
• easier to automate manufacturing
Disadvantages of SMT

- more heat generated
- small clearance makes cleaning difficult
- visual inspection difficult
- good joint formation important for mechanical reliability of assembly
- harder to hand assemble
- greater number of different materials to match CTE’s
Printed Circuit Boards (PCBs)

Most commonly encountered types of substrates:

– Laminates (FR-4, etc.)
– Ceramics
– Flex

For more information, see *High Performance Printed Circuit Boards* by Harper (McGraw-Hill)
FR-4 is the most widely used material because it’s adequate for most applications and cheap

When not to use FR-4:
- High reliability and/or hot components: high $T_g$, like FR-405, or even higher temp with ceramic
- High frequency: low dielectric loss ($\tan \delta$), such as PTFE (Teflon)
- High speed digital lower dielectric constants ($\varepsilon_r$), polyimide or PTFE
- Form factors: flex can turn corners
- Need CTE match to chip: ceramic
## Some PCB Laminate Materials

<table>
<thead>
<tr>
<th>NEMA Grade</th>
<th>Resin System</th>
<th>Reinforcement</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FR-2</td>
<td>Phenolic</td>
<td>Paper</td>
<td>Punchable, flame resistant</td>
</tr>
<tr>
<td>FR-3</td>
<td>Epoxy</td>
<td>Paper</td>
<td>Flame resistant, high insulation resistance</td>
</tr>
<tr>
<td>FR-4</td>
<td>Epoxy</td>
<td>Woven glass</td>
<td>Flame resistant, higher Tg, better thermal</td>
</tr>
<tr>
<td>FR-5</td>
<td>Epoxy</td>
<td>Woven glass</td>
<td>Flame resistant, low capacitance or high impact applications</td>
</tr>
<tr>
<td>FR-6</td>
<td>Polyester</td>
<td>Glass matte</td>
<td>Flame resistant, low capacitance or high impact applications</td>
</tr>
<tr>
<td>CEM-1</td>
<td>Epoxy</td>
<td>Paper and glass</td>
<td>Paper core and glass surface, self-extinguishing, excellent punching, longer drill life and minimal dust.</td>
</tr>
<tr>
<td>CEM-3</td>
<td>Epoxy</td>
<td>Woven glass and glass matte</td>
<td>Nonwoven glass core and woven glass surface, similar to FR-4, longer drill life</td>
</tr>
</tbody>
</table>
How to make PCBs

- Make (buy) FR4 laminate core
- Pattern Cu
- Laminate (press and heat)
- Drill
- Plate Cu
- Route images
- Test
How Laminates are Made

1. Roll of woven glass
2. Impregnate glass with epoxy resin
3. Dry/Cure
4. Cut
5. FR4 core laminate

Prepreg: semicured material that is dry and nontacky. It can be stored.

Press

Copper Foil + Prepreg
How PCBs are Made

- FR4 laminate core
- Pattern Cu
- Layer with prepreg and laminate (press and heat)
- Drill (plate outer layer and holes)
- Pattern outer layer (Route images & test)
Use a layout program to do design, component placement, and footprint definition:

- Cadence’s Allegro or Orcad
- Pads/Innoveda’s PowerPCB
- Mentor’s Board Station
- Protel
- others
• Design libraries are available for most parts
• New footprints can be added manually
• Often footprints can be downloaded from the part vendor or from Topline (http://www.toplinedummy.com)

• There are IPC design guidelines (IPC-SM-782 at http://www.ipc.org) and Jede component definitions (http://www.jedec.org)

In prototypes, you’re most concerned with fitting the part on the board properly, but in real products we consider joint geometry for manufacturing yield and product reliability. (footprint = pad dimensions and land patterns)
How to Specify PCBs

This is the information you should provide when ordering PCBs:

1. Quantity and lead time
2. X-Y dimensions/boards per panel, number of sides with components
3. Board material, thickness (4 layer boards usually 0.062”) and tolerances
4. Layer count and copper weight for layers:
   - ½ oz or 1oz copper on outer layers (less copper means shorter etch times)
   - 1 oz copper on inner layers (carry more current for ground/power planes)
5. Metallization (SnPb/HASL, organic, Cu-Ni-Au, immersion Sn or Ag or Au)
6. Minimum line and space width (< 0.008” costs more)
7. Hole count, min hole dim and finish (holes < 0.015” cost more)
8. Surface mount pad count and minimum pad pitch
9. Silkscreen and solder mask (usually green LPI)
10. Electrical testing requirements (need netlist for electrical test)
11. Gerber data (always create a README file)
Common SMT components

- QFP, SOIC, TSOP (gull wing)
- area array (BGA, CSP, flip chip)

May not be available as surface mount:
- Some connectors
- Transformers/solenoids
- Large electrolytic caps

- chip resistors, capacitors
- small outline transistors (SOT)
- PLCC (J lead)
Ordering SMT Components

For small numbers of parts (prototype quantities), use component distributors, such as:

- Digi-Key [http://www.digikey.com](http://www.digikey.com)
- Newark [http://www.newark.com](http://www.newark.com)
- Keytronics [http://www.keytronics.com](http://www.keytronics.com)
- Avnet [http://www.avnet.com](http://www.avnet.com)
- Jameco [http://www.jameco.com](http://www.jameco.com)
- EDX [http://www.edxelectronics.com](http://www.edxelectronics.com)

Etc., etc., etc.

Online ordering is easy. Look around for good prices.
Specifying SMT Components

Components are usually ordered by part number. Make sure you have the correct:

- Functional specs and tolerances
- Package type (QFP, TSOP, etc.)
- Lead type (gull wing, J-lead, etc.)
- X-Y dimensions (e.g. TSOPs can have the same number of pins but different body lengths and widths)
- Pins/pin outs/footprint
- Bulk packaging (tape & reel, tubes, trays)
- Quantity

for the part number you request.

Ordering more is cheaper per part, but don’t order parts you won’t use.
Assembly

Surface mount assembly process steps:

• Solder paste printing or dispensing
• Component placement
• Reflow
• Inspection
• Rework/backload
• Cleaning

A good reference: *Surface Mount Technology* by Prasad (ITP)
Solder paste has tiny metal spheres of the alloy mixed with flux, solvents, and thixotropic materials.

Methods of applying solder paste:
- Stencil printing
- Syringe dispensing

Most influential step affecting yield
Reflow

Once parts have been placed on the solder paste bricks, the entire board is placed in an oven and taken through a temperature profile like:
- Look for wrong/misplaced components and poor solder joints
- Fix problems and add parts that can't survive the high temperature of the reflow oven
- Wash to remove flux residues
Assembly - yourself

Use large components / large pitch
Dispense (usually SnPb solder paste)
- Use a robust paste with a wide process window
  - Alpha WS609 (if you can clean the board or don’t care about long term reliability)
  - Kester R244 if you can’t clean
Hand place components with tweezers
- don’t let paste dry out
- don’t push down too hard
- always use ESD protection
Hot plate
- only needs to be molten (~200°C) for 60-90s
Clean, if necessary
Rework and hand soldering

unreflowed solder paste

Defects happen in the best manufacturing process:

• Wrong part
• Reversed polarity
• Misaligned part
• Shorts/bridging/excess solder
• Opens/insufficient solder
• Nonwetting/unreflowed solder
Rework

Remove component
Clean pads
Re-tin pads
Install new component
Removing Components (using hot air solder system)

1. Applying flux to all land/leaded areas
2. Position the nozzle over part
3. Turn on vacuum and set vacuum cup on part
4. Lower nozzle and melt all joints
5. Lift component
Remove Old Solder
(with blade tip on soldering iron)

1. Apply flux to lands

2. Lay braid on solder to be removed

3. Place iron tip on braid, and when solder flow stops, remove braid and tip
Re-tin and Level Pads (with blade tip on soldering iron)

- Apply flux to lands
- Tin the blade tip
- Place the blade lightly along the center line of the row of lands
- Gently draw the tip off the lands after the solder melts
Install New Component
(using hot air pencil)

1. Dispense solder paste in a long, single line over pads

2. Place component

3. Adjust air pressure

4. Dry paste until it appears dull

5. Move tip closer and heat until solder melts

6. Clean, if necessary
Fixing Shorts

1. Apply flux to the bridged leads
2. Clean tip of soldering iron
3. Hold the tip so that it runs parallel to the row of leads
4. Bring the flat surface of the tip down on the bridge and wait for reflow
5. Draw the bridge gently down away from the component

Figure 1  Apply Flux
Figure 2  Hold Tip Parallel
Figure 3  Draw Tip Away From Component
1. Apply flux to open lead
2. Used flux cored solder wire to apply tin to the soldering tip
3. Bring the tip in at a 45° angle and make contact with lead and land where they meet
4. Draw the tip away