NPTEL VIDEO COURSE ON ELECTRONICS SYSTEMS PACKAGING

- Introduction to systems packaging
- Semiconductor packaging overview and fundamentals
- Packages
- Electrical design issues in packaging
- CAD for printed wiring boards and Design for manufacturability
- Technology of system-level Printed Wiring Boards
A QUICK REVIEW OF THE CAD PROCESS STEPS BY ILLUSTRATIONS
Components that will be used in the electrical schematic are selected and designed.
The electrical components are placed in the schematic and net connections established.
The Bill of Materials is derived from the components that exist in the schematic.

<table>
<thead>
<tr>
<th>Item</th>
<th>Quantity</th>
<th>Referencenumber</th>
<th>Part</th>
<th>Value</th>
<th>Decal</th>
<th>MFG Part Number</th>
<th>Datasheet</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>C1</td>
<td>1000FF</td>
<td>+.1% 50V</td>
<td>1206</td>
<td>GRM251R470MA02G</td>
<td>\WurataCerCap.pdf</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>C2</td>
<td>01UF0D</td>
<td>+.1% 50V</td>
<td>1206</td>
<td>GRM251R470MA02G</td>
<td>\WurataCerCap.pdf</td>
</tr>
<tr>
<td>3</td>
<td>14</td>
<td>C4</td>
<td>470FD</td>
<td>+.1% 50V</td>
<td>1206</td>
<td>GRM251R470MA02G</td>
<td>\WurataCerCap.pdf</td>
</tr>
<tr>
<td>4</td>
<td>5</td>
<td>C5</td>
<td>33uf 25V</td>
<td>+.1% 25V</td>
<td>1206</td>
<td>GRM251R470MA02G</td>
<td>\WurataCerCap.pdf</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>C6</td>
<td>33uf 25V</td>
<td>+.1% 25V</td>
<td>1206</td>
<td>GRM251R470MA02G</td>
<td>\WurataCerCap.pdf</td>
</tr>
<tr>
<td>6</td>
<td>5</td>
<td>C7</td>
<td>1UF0D</td>
<td>+.1% 50V</td>
<td>1206</td>
<td>GRM251R470MA02G</td>
<td>\WurataCerCap.pdf</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>C8</td>
<td>1UF0D</td>
<td>+.1% 50V</td>
<td>1206</td>
<td>GRM251R470MA02G</td>
<td>\WurataCerCap.pdf</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>C9</td>
<td>1UF0D</td>
<td>+.1% 50V</td>
<td>1206</td>
<td>GRM251R470MA02G</td>
<td>\WurataCerCap.pdf</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>C10</td>
<td>01UF0D</td>
<td>+.1% 50V</td>
<td>1206</td>
<td>GRM251R470MA02G</td>
<td>\WurataCerCap.pdf</td>
</tr>
<tr>
<td>10</td>
<td>6</td>
<td>D1</td>
<td>MBR5120T3</td>
<td>-</td>
<td>403A</td>
<td>MBR5120T3</td>
<td>\MBR5120T3.pdf</td>
</tr>
<tr>
<td>11</td>
<td>1</td>
<td>D2</td>
<td>MRA4000T3</td>
<td>-</td>
<td>403D</td>
<td>MRA4000T3</td>
<td>\MRA4000T3.pdf</td>
</tr>
<tr>
<td>12</td>
<td>1</td>
<td>D3</td>
<td>MRA4000T3</td>
<td>-</td>
<td>403D</td>
<td>MRA4000T3</td>
<td>\MRA4000T3.pdf</td>
</tr>
<tr>
<td>13</td>
<td>10</td>
<td>D4</td>
<td>R1418</td>
<td>-</td>
<td>MELF3</td>
<td>L0146</td>
<td>\R1418.pdf</td>
</tr>
<tr>
<td>14</td>
<td>10</td>
<td>D5</td>
<td>R1418</td>
<td>-</td>
<td>MELF3</td>
<td>L0146</td>
<td>\R1418.pdf</td>
</tr>
<tr>
<td>15</td>
<td>10</td>
<td>D6</td>
<td>R1418</td>
<td>-</td>
<td>MELF3</td>
<td>L0146</td>
<td>\R1418.pdf</td>
</tr>
<tr>
<td>16</td>
<td>10</td>
<td>D7</td>
<td>R1418</td>
<td>-</td>
<td>MELF3</td>
<td>L0146</td>
<td>\R1418.pdf</td>
</tr>
<tr>
<td>17</td>
<td>10</td>
<td>D8</td>
<td>R1418</td>
<td>-</td>
<td>MELF3</td>
<td>L0146</td>
<td>\R1418.pdf</td>
</tr>
<tr>
<td>18</td>
<td>10</td>
<td>D9</td>
<td>R1418</td>
<td>-</td>
<td>MELF3</td>
<td>L0146</td>
<td>\R1418.pdf</td>
</tr>
<tr>
<td>19</td>
<td>1</td>
<td>D10</td>
<td>SMIC16A</td>
<td>-</td>
<td>DO-214AB</td>
<td>SMIC16A</td>
<td>\temp.pdf</td>
</tr>
</tbody>
</table>

Courtesy: PCB3D.COM
The components that reside on the pcb are designed from the Bill of Materials. These pcb library parts are captured in the netlist, and consist of a decal and a part name.
The netlist is typically an ASCII format generated from the schematic. It contains all components (*part*) and connections (*net*) required for the pcb design.

---

*Courtesy: PCB3D.COM*
The netlist is imported into the pcb database. If all components and connections from the netlist match the pcb database of library components, they will appear as pictured below. Green indicates all components (*part*), yellow indicates connections (*net*) to the components.

*Courtesy PCBD.COM*
Components from the netlist are dispersed and grouped according to function. The components are then manually placed inside the pcb outline.
PCB Design: Placement

Components are placed within the pcb board outline. Keepouts, cutouts and holes must be avoided.
PCB Design: Routing

All connections (nets) require trace routing.
The red lines are completed trace connections.
The yellow lines represent unrouted traces or nets.
Plane connections are created with a via or direct connection to a copper area.
Trace routing may consist of component to component or component to plane connections.
Copper areas are created (red) and poured over vias (white circles) and solder pads (violet). Copper areas are then assigned a net name that matches with the appropriate net connection.

Courtesy: PCB3D.COM
The copper planes are created, split and defined according to the design rules and net requirements for each layer. Thermal and non-thermal connections are placed accordingly.
Design for Test involves placement of test points into the completed PCB. Two types of test points are used: Flying Probe (FPT) and In Circuit (ICT). DFT analysis is executed and test points are audited for compliance and testability.
PCB Design: Verification

The PCB is complete and must be verified for design rules compliance. Verification includes clearance checks, net and copper connectivity, net and copper plane connectivity, duplicate nets, layer to layer connectivity, design rule violations, DFM and test points.

In this example, the yellow circles indicate a clearance violation (copper to board edge).

Courtesy: PC33D.COM
Gerber Files

Gerber files are created to enable plotting of the individual design file elements. Depending on their function, each Gerber file is compiled as an individual electrical layer, process or design reference.

Typical Gerber File Structure

- Electrical Design Layers
- Silkscreen
- Solder Mask
- Solder Paste
- Fabrication Drawings
- Assembly Drawings
- Aperture Files
- Drill Files
- Netlist
- X-Y Placement Data

Courtesy: PCB3D.COM
Gerber Files: Electrical Layers

These Gerber files are processed to create each electrical layer (internal and external) that will ultimately be finished in copper on the pcb.

Courtesy: PCB3D.COM
Gerber Files: Silkscreen

This file will create the stencil that will be used to apply the silkscreen (ink) to the pcb. The Silkscreen is for component reference, identification and labeling. The Silkscreen exists on the outer layers.
The solder mask will expose solderable areas and protect the pcb by covering all copper elements. The Solder Mask minimizes solder bridges. In this example, the Gerber file was created as a negative. Areas in red will not be covered with mask. The solder mask exists on the outer layers.
Gerber Files: Solder Paste

This file will be used to create a solder paste stencil. Prior to the board assembly, the stencil will be used to apply solder paste directly to the pads on the pcb (areas in yellow). Once the solder paste is applied, surface mount components can be placed and soldered.

Courtesy: PCB3D.COM
Gerber Files: Assembly Drawing

Identifies location and orientation of the electronic components to be placed.
This listing defines the shape of individual elements on the pcb.

<table>
<thead>
<tr>
<th>POSITION</th>
<th>WHICH</th>
<th>H/D/ID</th>
<th>SHAPE</th>
<th>REY</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>23</td>
<td>0</td>
<td>RND</td>
<td>292</td>
</tr>
<tr>
<td>11</td>
<td>28</td>
<td>0</td>
<td>RND</td>
<td>95</td>
</tr>
<tr>
<td>12</td>
<td>10</td>
<td>0</td>
<td>RND</td>
<td>3906</td>
</tr>
<tr>
<td>13</td>
<td>7</td>
<td>0</td>
<td>RND</td>
<td>09</td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>0</td>
<td>RND</td>
<td>5650</td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>0</td>
<td>RND</td>
<td>4011</td>
</tr>
<tr>
<td>16</td>
<td>64</td>
<td>0</td>
<td>RND</td>
<td>8</td>
</tr>
<tr>
<td>17</td>
<td>70</td>
<td>0</td>
<td>RND</td>
<td>28</td>
</tr>
<tr>
<td>18</td>
<td>45</td>
<td>70</td>
<td>RECT</td>
<td>38</td>
</tr>
<tr>
<td>19</td>
<td>30</td>
<td>0</td>
<td>RND</td>
<td>805</td>
</tr>
<tr>
<td>20</td>
<td>50</td>
<td>0</td>
<td>SQR</td>
<td>38</td>
</tr>
<tr>
<td>21</td>
<td>45</td>
<td>0</td>
<td>SQR</td>
<td>22</td>
</tr>
<tr>
<td>22</td>
<td>17</td>
<td>0</td>
<td>RND</td>
<td>137</td>
</tr>
<tr>
<td>23</td>
<td>13</td>
<td>0</td>
<td>RND</td>
<td>40</td>
</tr>
<tr>
<td>24</td>
<td>55</td>
<td>0</td>
<td>RND</td>
<td>187</td>
</tr>
<tr>
<td>25</td>
<td>12</td>
<td>0</td>
<td>RND</td>
<td>37</td>
</tr>
<tr>
<td>26</td>
<td>1</td>
<td>0</td>
<td>RND</td>
<td>1292</td>
</tr>
<tr>
<td>27</td>
<td>102</td>
<td>0</td>
<td>RND</td>
<td>16</td>
</tr>
<tr>
<td>28</td>
<td>75</td>
<td>0</td>
<td>RND</td>
<td>8</td>
</tr>
<tr>
<td>29</td>
<td>55</td>
<td>0</td>
<td>SQR</td>
<td>28</td>
</tr>
<tr>
<td>30</td>
<td>101</td>
<td>0</td>
<td>SQR</td>
<td>10</td>
</tr>
<tr>
<td>31</td>
<td>84</td>
<td>0</td>
<td>SQR</td>
<td>90</td>
</tr>
<tr>
<td>32</td>
<td>27</td>
<td>0</td>
<td>SQR</td>
<td>40</td>
</tr>
<tr>
<td>33</td>
<td>39</td>
<td>0</td>
<td>SQR</td>
<td>1</td>
</tr>
<tr>
<td>34</td>
<td>0.3</td>
<td>0</td>
<td>RND</td>
<td>28</td>
</tr>
<tr>
<td>35</td>
<td>4</td>
<td>0</td>
<td>RND</td>
<td>441</td>
</tr>
<tr>
<td>36</td>
<td>5</td>
<td>0</td>
<td>RND</td>
<td>48</td>
</tr>
<tr>
<td>37</td>
<td>30</td>
<td>0</td>
<td>SQR</td>
<td>26</td>
</tr>
<tr>
<td>38</td>
<td>85</td>
<td>0</td>
<td>SQR</td>
<td>10</td>
</tr>
<tr>
<td>39</td>
<td>0.3</td>
<td>0</td>
<td>SQR</td>
<td>100</td>
</tr>
<tr>
<td>40</td>
<td>0.3</td>
<td>0</td>
<td>SQR</td>
<td>71</td>
</tr>
<tr>
<td>41</td>
<td>25</td>
<td>0</td>
<td>SQR</td>
<td>29</td>
</tr>
<tr>
<td>42</td>
<td>18</td>
<td>0</td>
<td>SQR</td>
<td>272</td>
</tr>
<tr>
<td>43</td>
<td>27</td>
<td>0</td>
<td>SQR</td>
<td>36</td>
</tr>
<tr>
<td>44</td>
<td>15</td>
<td>0</td>
<td>SQR</td>
<td>7</td>
</tr>
<tr>
<td>45</td>
<td>25</td>
<td>0</td>
<td>SQR</td>
<td>24</td>
</tr>
<tr>
<td>46</td>
<td>45</td>
<td>0</td>
<td>SQR</td>
<td>84</td>
</tr>
<tr>
<td>47</td>
<td>35</td>
<td>0</td>
<td>SQR</td>
<td>10</td>
</tr>
</tbody>
</table>
Gerber Files: Drill Files

CNC drill parameters used on the pcb fabricators system to drill and route the pcb.

Courtesy: PCB3D.COM
PCB Fabrication

PCB Basics

Basic PCBs comprise a rigid sheet of epoxy-impregnated fiberglass material within copper sheets affixed to one or both sides. This is known as copper clad. In multilayer boards (those with more than two copper layers), a piece of material called prepreg is placed between core layers.

Courtesy: PCB3D.COM
NEXT CHAPTER:

**PRINTED WIRING BOARD TECHNOLOGIES**
Photo tool (mask) generation

- Artwork, photomaster, phototool, mask
- CAD output in Gerber format (universal)
- X,Y table with light source
- Scanning, editing Gerber files
- Raster, Vector plotting equipment
- 4mil and 7mil Ag-halide film
- Photographic process
- Film stabilization
- D-code aperture listing
  Laser - Raster
The board can be GOOD if the Phototool is good
(minus Manufacturing tolerances)
QUALITY CONTROL IS CRUCIAL TO CHECK TOLERANCE LEVELS
Photoplotting equipment

- Photo plot on Silver halide films
- Use either 4 mil or 7 mil films but 7mil is preferred
- Gerber files are input data
- AutoCAD drawings are accepted
- Lithographic film used
- Large film sizes used e.g. 32” x 28”
- Drum type; vacuum hold on film
- Multiple plots can be done
- Plots can be fitted to PCB panel size
- Scan and Edit of tech files possible
- Chemical processing follows plotting
- Stabilization in controlled conditions for at least 4-6h
- Laser plotters (raster) cost about Rs. 70 lakhs
- Resolution as high as 40000dpi
- Accuracy (positioning): +/- 2-4um
- Min line width 35 um; 25 um currently expensive
- Negative and positive masks can be plotted
- Other masks- for Solder mask, silk screen
- Use red or mild-green safe lights for loading film
- Data transfer Protocol: Gerber, RS 274-X
- Light sources- different;
  - Xenon-LCD Image projection technology
- Laser Direct Imaging is now becoming accepted
  Requires a compatible photoresist

Fig. source: everythingpcb.com
Structure of Photo film

1. Protective Layer
2. Emulsion [Silver Halide in Gelatin]
3. Sub layer [Adhesion Promoter]
4. Base [Polyester film]
5. Sub Layer
6. Backing
7. Light absorbing chemicals

Total thickness
100, 175 microns
Silver halide photo tool

handle carefully

Possible Defects
- Pin holes ✓
- Scratches ✓
- Lift-off of flakes ✓
- Inadequate contrast Black & White
- No flatness

Di-azo films are used on shop floor
Exposure

Developing

- Latent image made visible
  - Metol or Hydroquinone
- Reducing action to metallic silver
  - Time and agitation important; concn. of solution
- Orthochromatic or "Lith" films used
  - Commercial term
- Safe lights for processing - red safe lights

Stopper bath - 1% acetic acid solution

Fixer bath

- Sodium thiosulfate; permanently 'fixes' the image

Wash, dry and stabilize

Dimensional stability required; 4mil and 7mil films

Diazo films - dry developing using ammonia

- Exposed areas turn amber color on ammonia
- Does cut off UV light as black silver halide - better registration
- Available as 7mil film only - better handling in shop floor
- Can be used for making multiple copies to avoid errors

Recycling?
Basic Steps in Manufacture
Single sided board

- Design
- Photo-tooling (1:1)
- Image/Print
- Etch (Unwanted Cu)
- Drill holes for component mounting
- Protect Cu (Solder)
- Solder mask
- Assemble (PTH)
Double sided board manufacture

- Design
- Photo-tooling (1:1)
- Drill holes (PTH)
- Plate (electroless)
- Image circuit
- Plate (Cu electroplate)
- Plate (Sn or Sn-Pb electroplate)
- Strip
- Etch
- Strip and Protect before assembly

Fig. source: Wikimedia Commons 2011