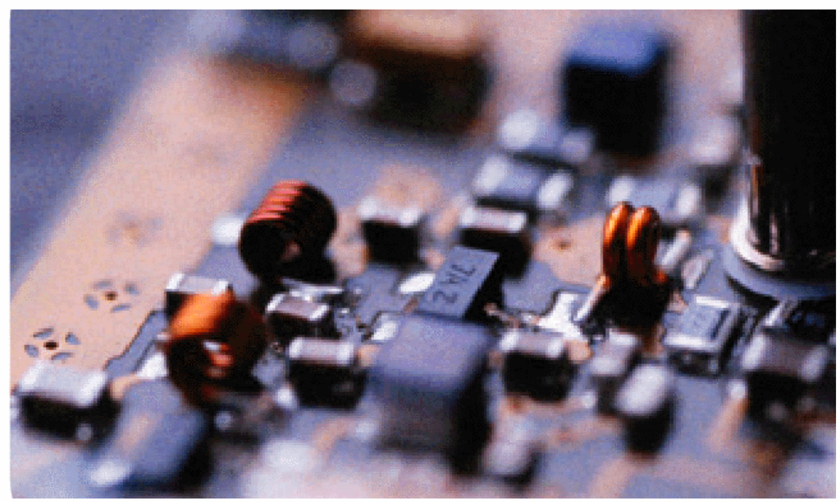
What is “surface mount”?

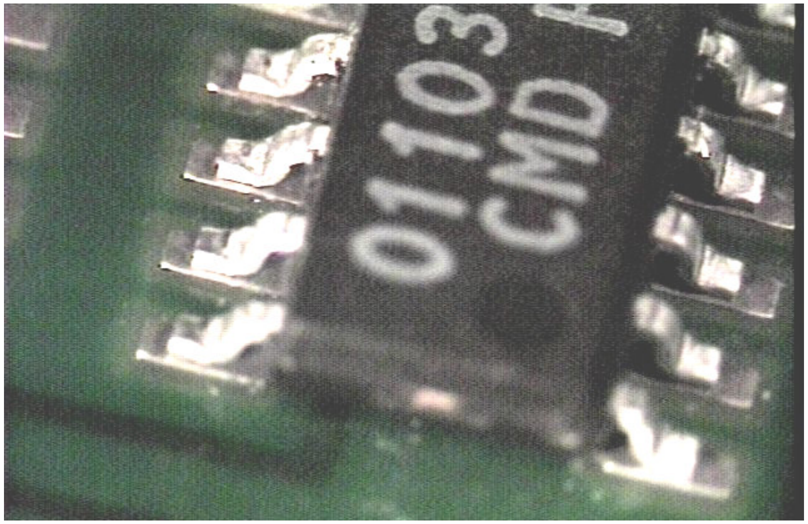
A way of attaching electronic components to a printed circuit board

The solder joint forms the mechanical and electrical connection

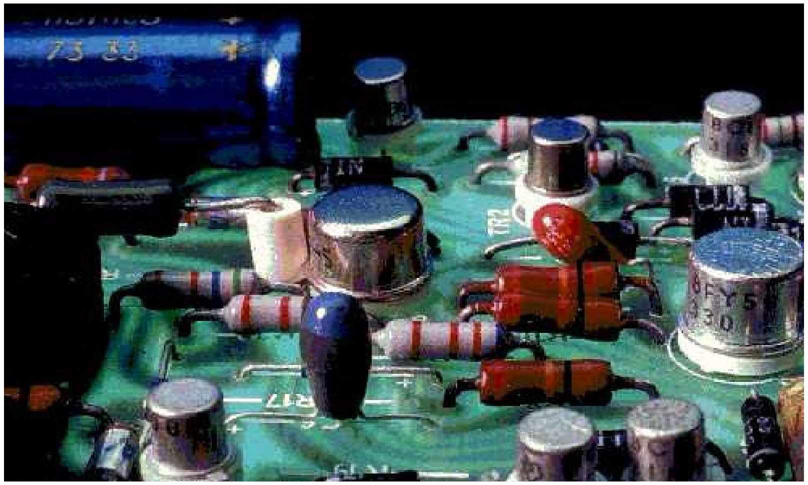
Bonding of the solder joint is to the surface of a conductive land pattern

Connection does not use through holes or terminals





Surface Mount



Through Hole

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

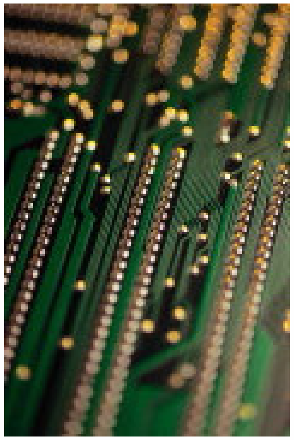
- more heat generated
- small clearance makes cleaning difficult
- visual inspection difficult
- good joint formation important for mechanical reliability of assembly
- harder to hand assemble
- greater number of different materials to match CTE’s

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

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 d), such as PTFE (Teflon)
- High speed digital lower dielectric constants (e_r), polyimide or PTFE
- Form factors: flex can turn corners
- Need CTE match to chip: ceramic

Some PCB Laminate Materials

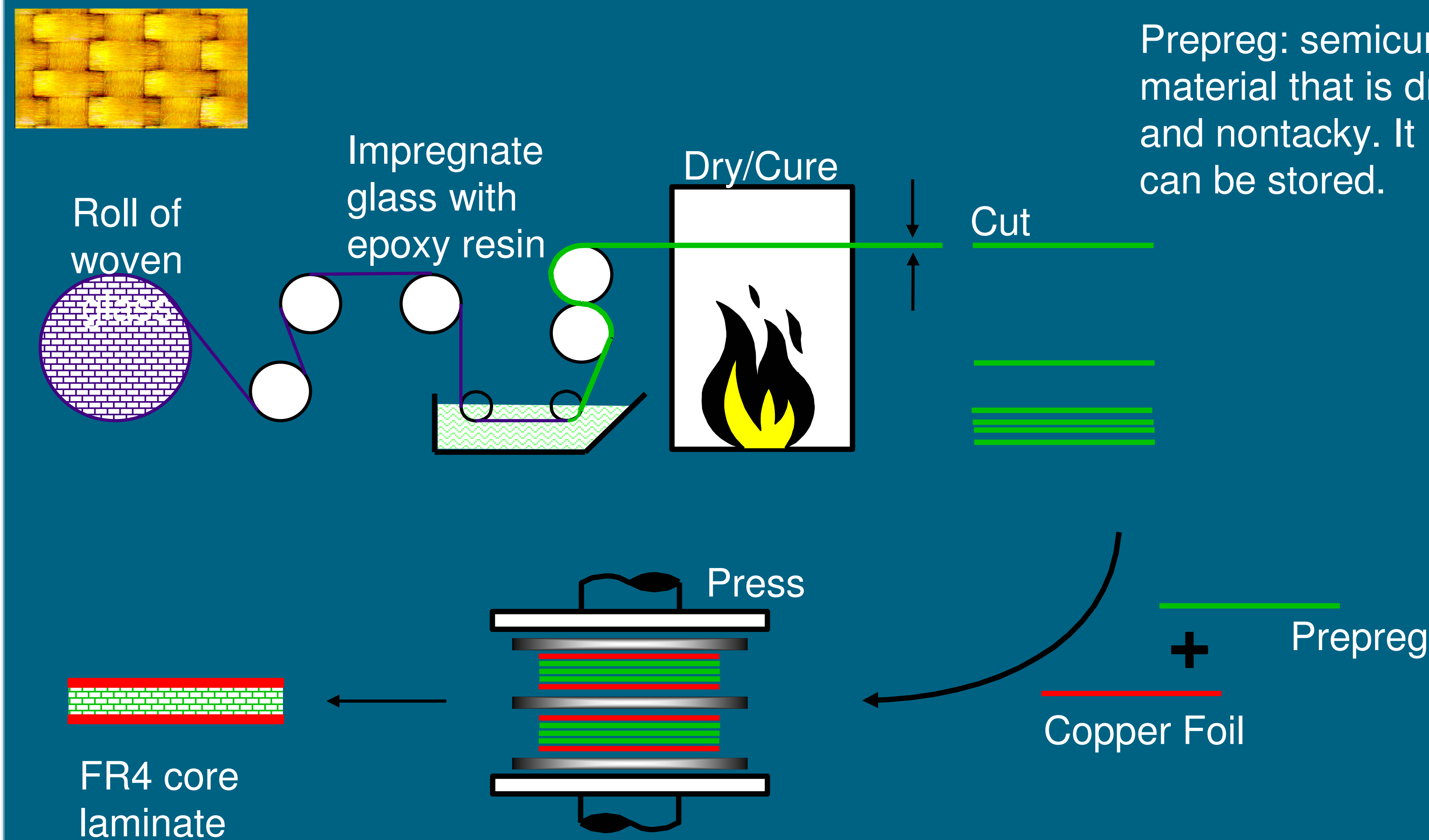
NEMA Grade	Resin System	Reinforcement	Description
FR-2	Phenolic	Paper	Punchable, flame resistant
FR-3	Epoxy	Paper	Flame resistant, high insulation resistance
FR-4	**Epoxy**	**Woven glass**	**Flame resistance, Tg ~ 130C**
FR-5	Epoxy	Woven glass	Flame resistant, higher Tg, better thermal
FR-6	Polyester	Glass matte	Flame resistant, low capacitance or high impact applications
CEM-1	Epoxy	Paper and glass	Paper core and glass surface, self-extinguishing, excellent punching, longer drill life and minimal dust.
CEM-3	Epoxy	Woven glass and glass matte	Nonwoven glass core and woven glass surface, similar to FR-4, longer drill life

How to make PCBs

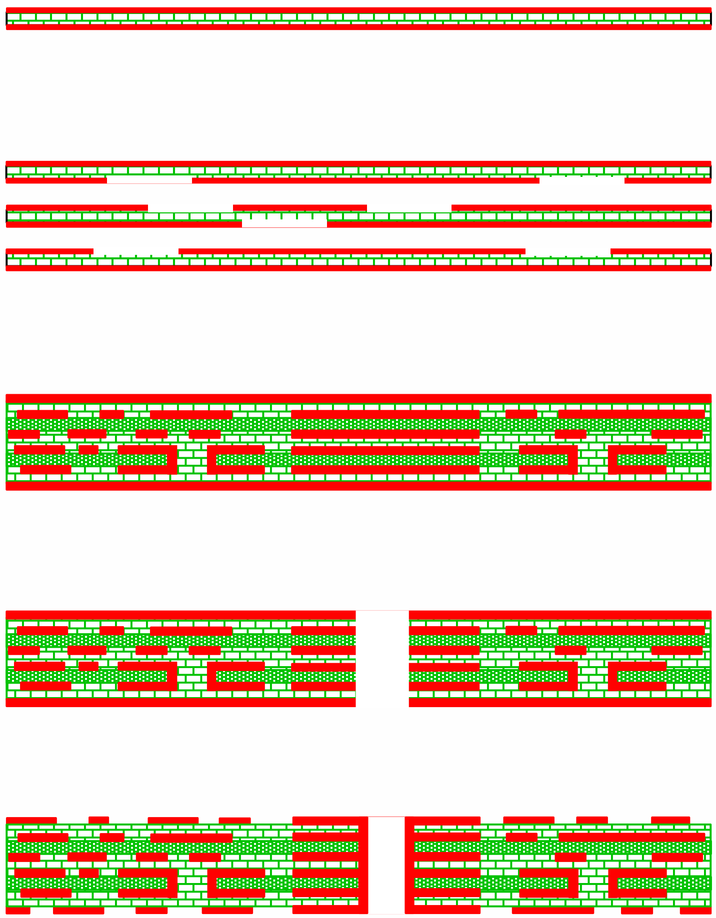
- Make (buy) FR4 laminate core
- Pattern Cu
- Laminate (press and heat)
- Drill
- Plate Cu
- Route images
- Test

How Laminates are Made

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



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)

SMT Layout

Use a layout program to do design, component placement, and footprint definition:

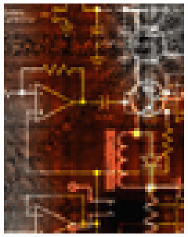
•Cadence's Allegro or Orcad

•Pads/Innoveda's PowerPCB

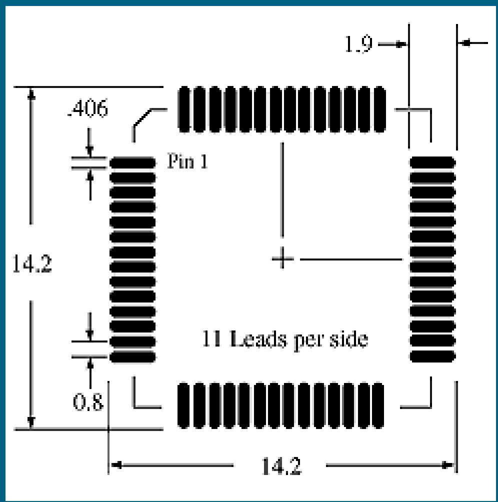
•Mentor's Board Station

•Protel

•others



Footprints



- 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 Jedec 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)

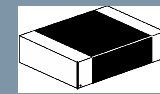
How to Specify PCBs

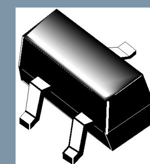
This is the information you should provide when ordering PCBs:

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

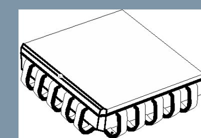
- chip resistors, capacitors



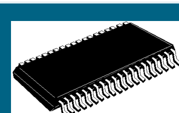


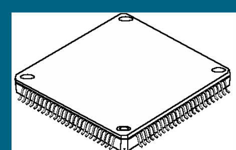
- small outline transistors (SOT)

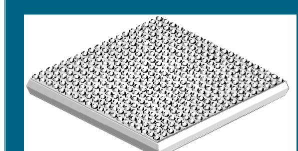
- PLCC (J lead)



- 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

Ordering SMT Components

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

- Digi-Key http://www.digikey.com
- Newark http://www.newark.com
- Keytronics http://www.keytronics.com
- Avnet http://www.avnet.com
- Jameco http://www.jameco.com
- EDX 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

<u>for the part number you request</u>.

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)

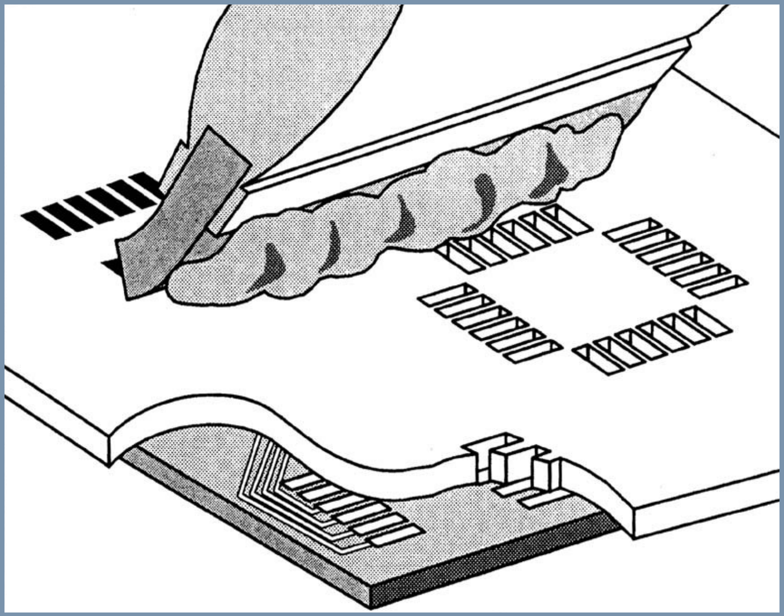
Paste and printing

Solder paste has tiny metal spheres of the alloy mixed with flux, solvents, and thixotropic materials

Methods of applying solder paste:

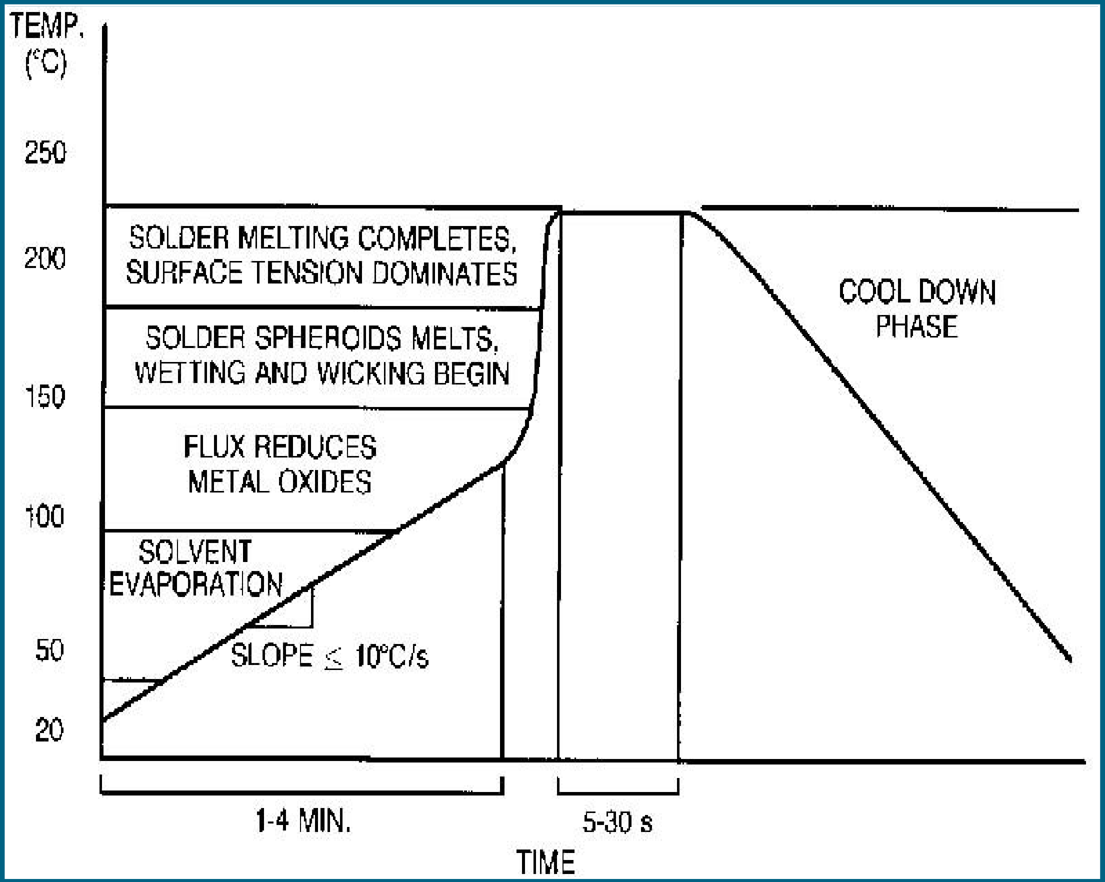
- Stencil printing
- Syringe dispensing

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:



Inspection/Test

- Look for wrong/misplaced components and poor solder joints

Rework/Backload

- Fix problems and add parts that can't survive the high temperature of the reflow oven

Cleaning

- 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

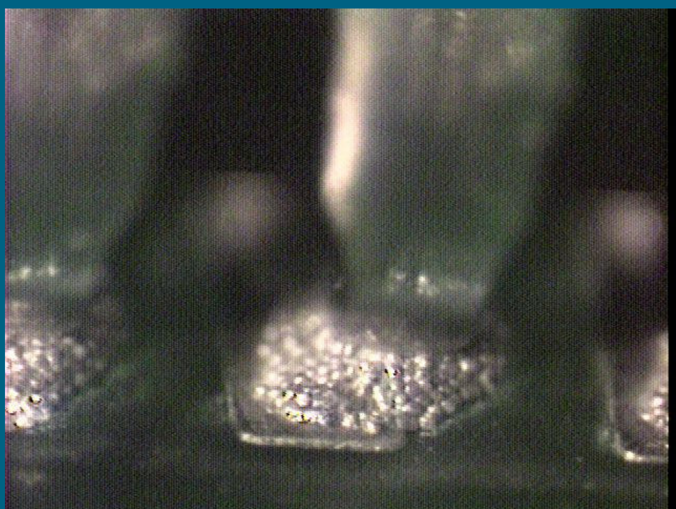
- don't let paste dry out
- don't push down too hard
- always use ESD protection

Hot plate

- only needs to be molten (~200C) 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 al
   land/leaded areas

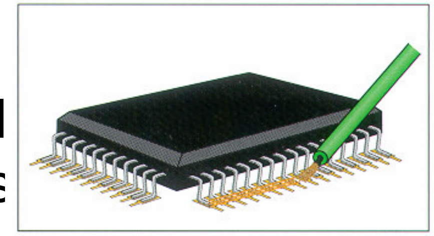


Figure 1 Flux Component

2. Position the
   nozzle over part

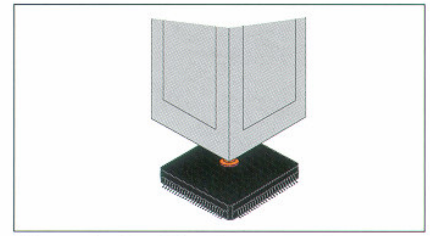


Figure 2 Position Component

3. Turn on vacuum
   and and set
   vacuum cup on
   part

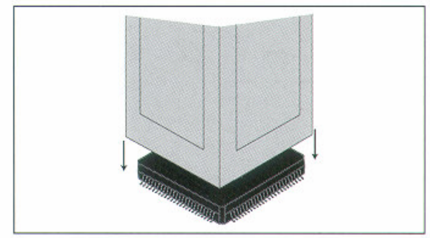


Figure 3 Lower Nozzle

4. Lower nozzle and
   melt all joints

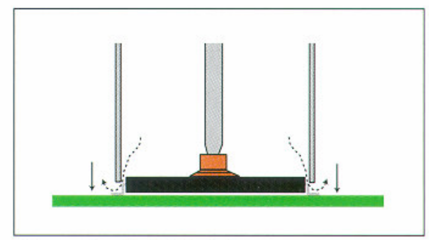


Figure 4 Melt All Joints

5. Lift component

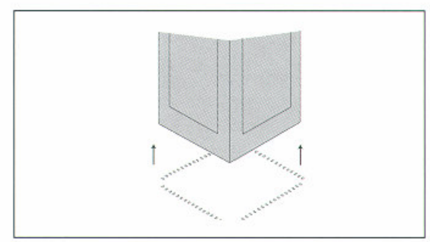


Figure 5 Lift Component

Remove Old Solder
(with blade tip on soldering iron)

1. Apply flux to lands

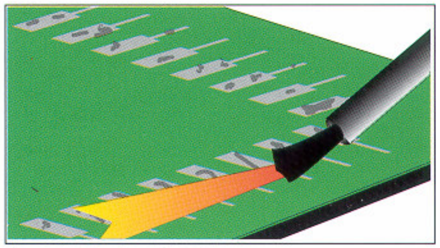


Figure 2 Apply Flux

2. Lay braid on solder to be removed

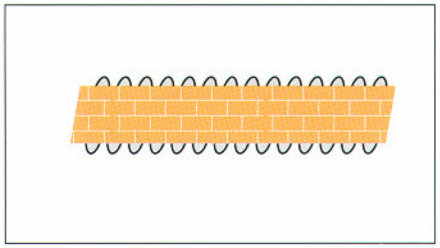


Figure 3

3. Place iron tip on braid, and when solder flow stops, remove braid and tip

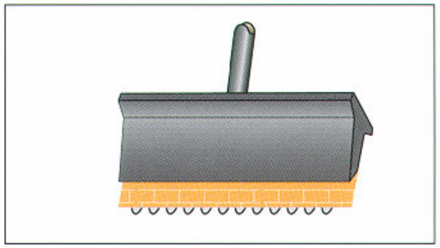
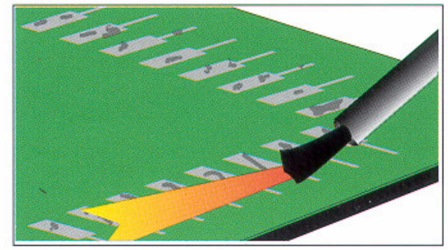


Figure 4

Re-tin and Level Pads
(with blade tip on soldering iron)

- Apply flux to lands



- Tin the blade tip

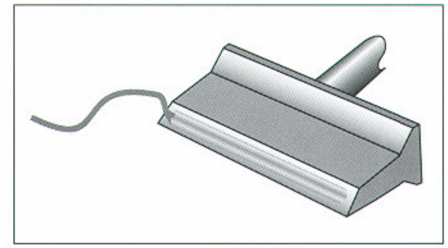


Figure 2 Tin Tip

- Place the blade lightly along the center line of the row of lands

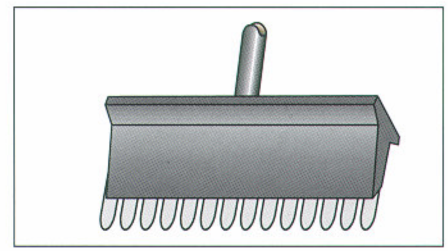


Figure 3 Place Beveled Edge Across

- Gently draw the tip off the lands after the solder melts

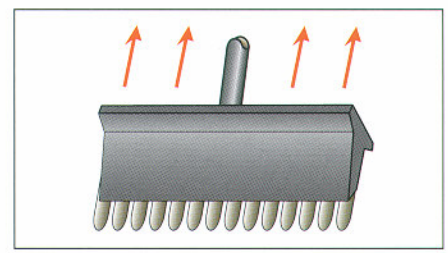


Figure 4 Draw Tip Off Lands

Install New Component
(using hot air pencil)

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

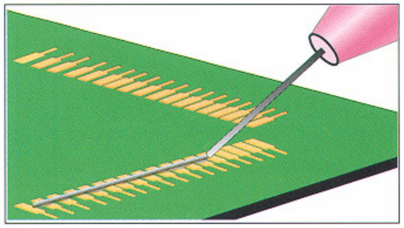


Figure 1 Apply Solder Paste

2. Place component

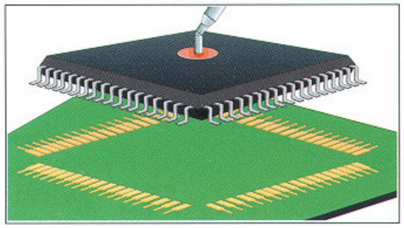


Figure 2 Position Component

3. Adjust air pressure

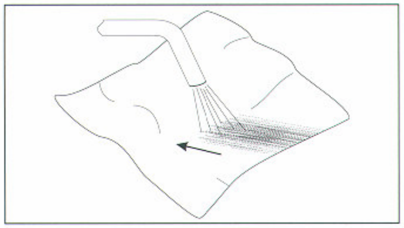


Figure 3 Adjust Pressure

4. Dry paste until it appears dull

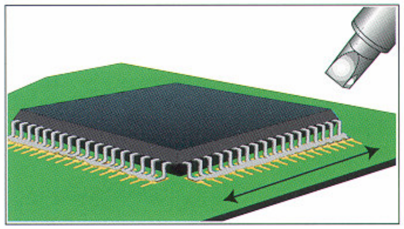


Figure 4 Pre-dry Paste

5. Move tip closer and heat until solder melts

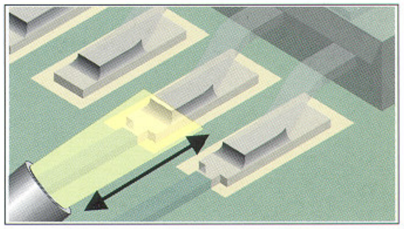


Figure 5 Melt Joints

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

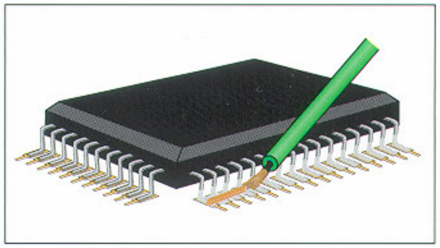


Figure 1 Apply Flux

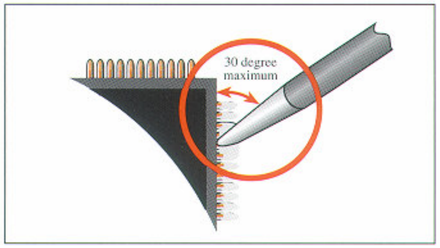


Figure 2 Hold Tip Parallel

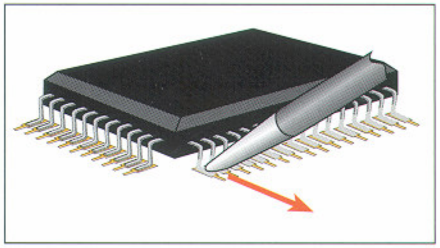


Figure 3 Draw Tip Away From Component

Fixing Opens

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^{o} angle and make contact with lead and land where they meet
4. Draw the tip away

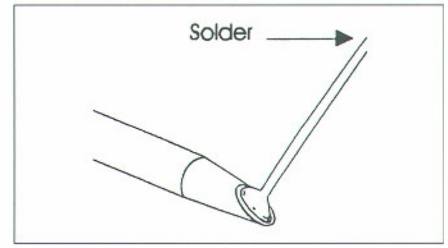


Figure 2 Apply Solder to Tip

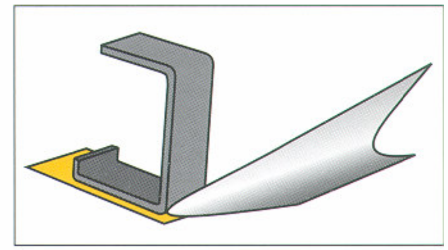
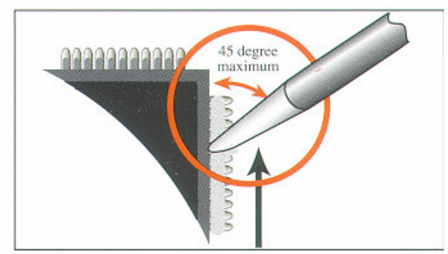
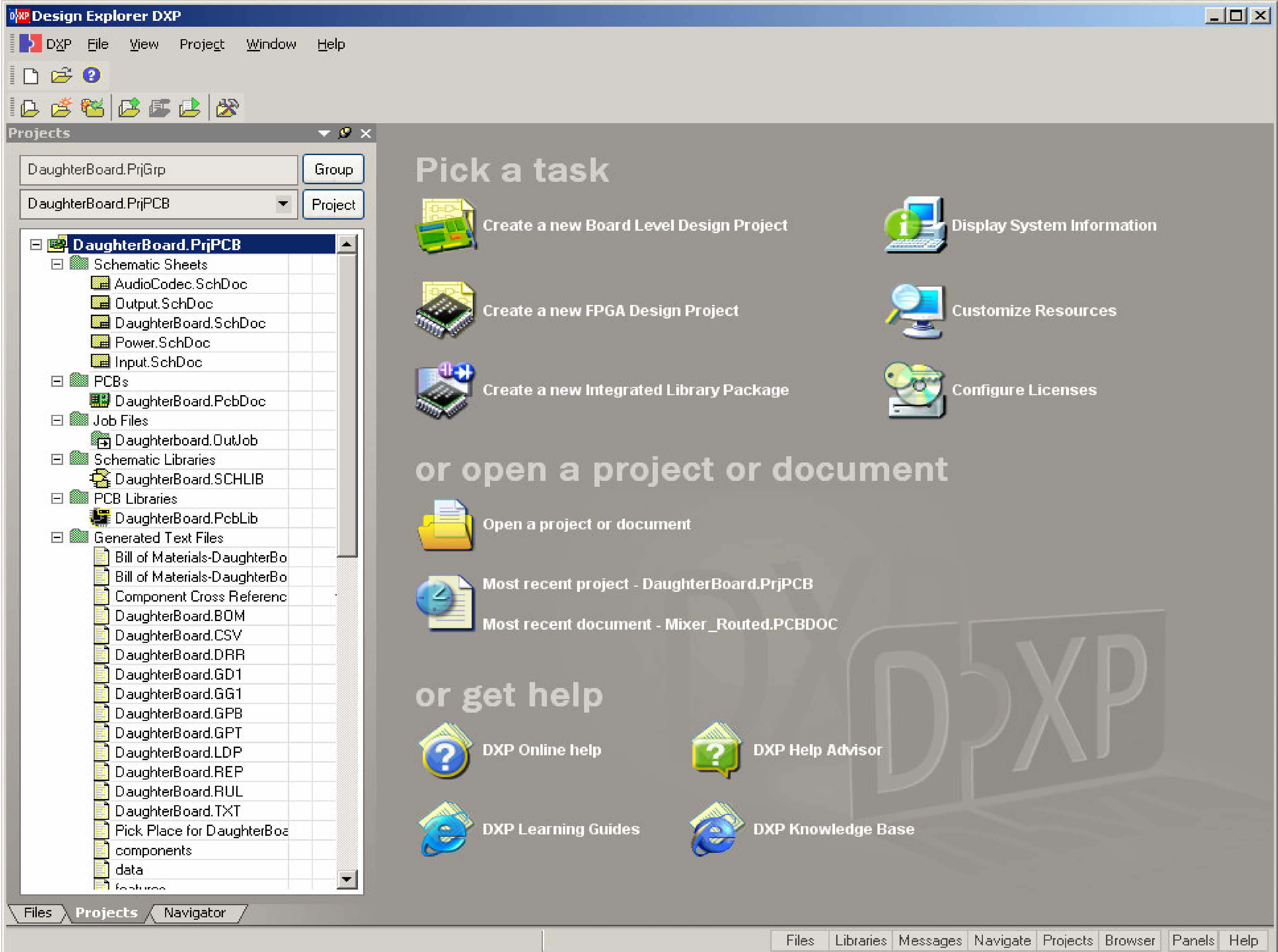


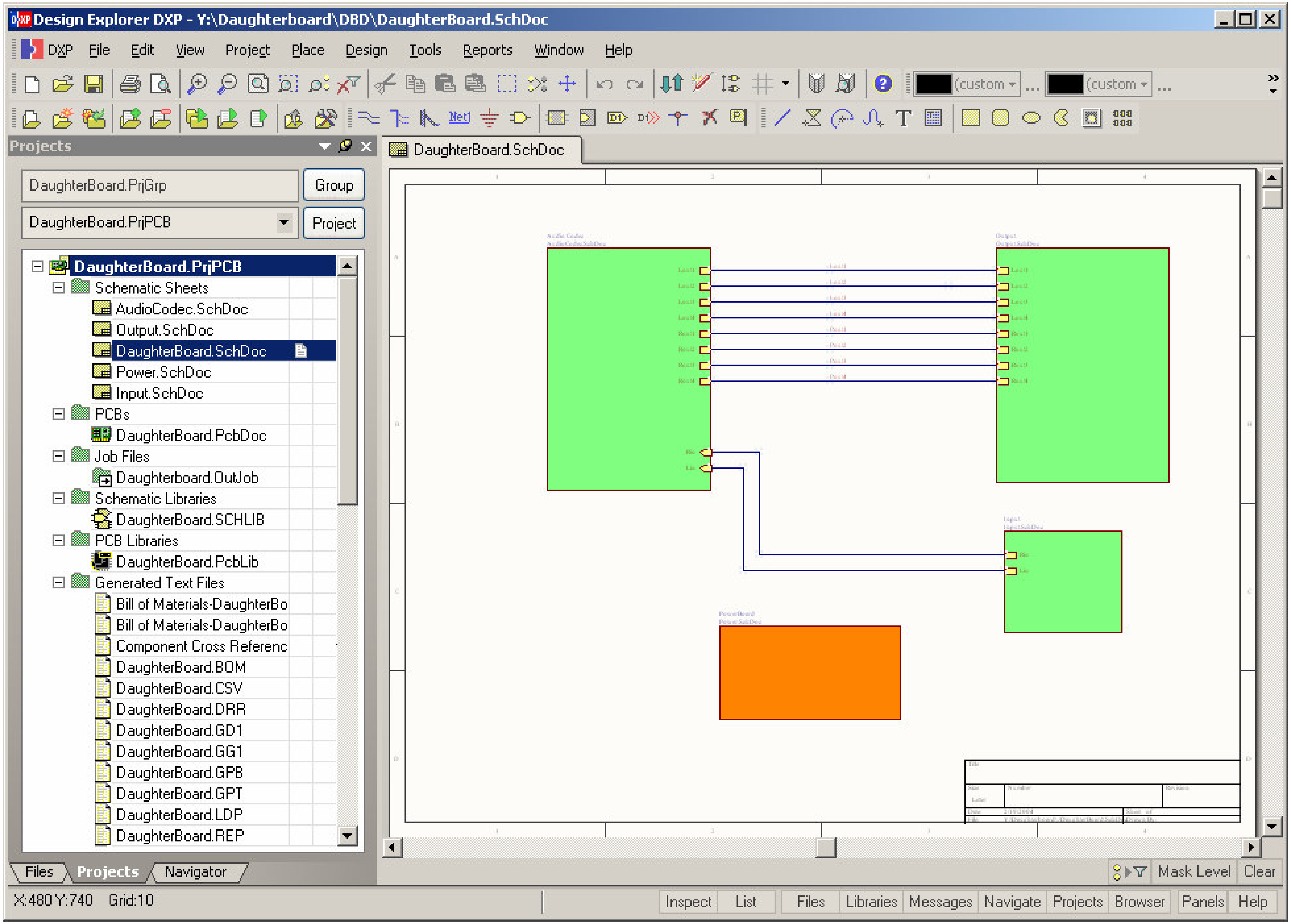
Figure 3 Tip in Contact with Heel

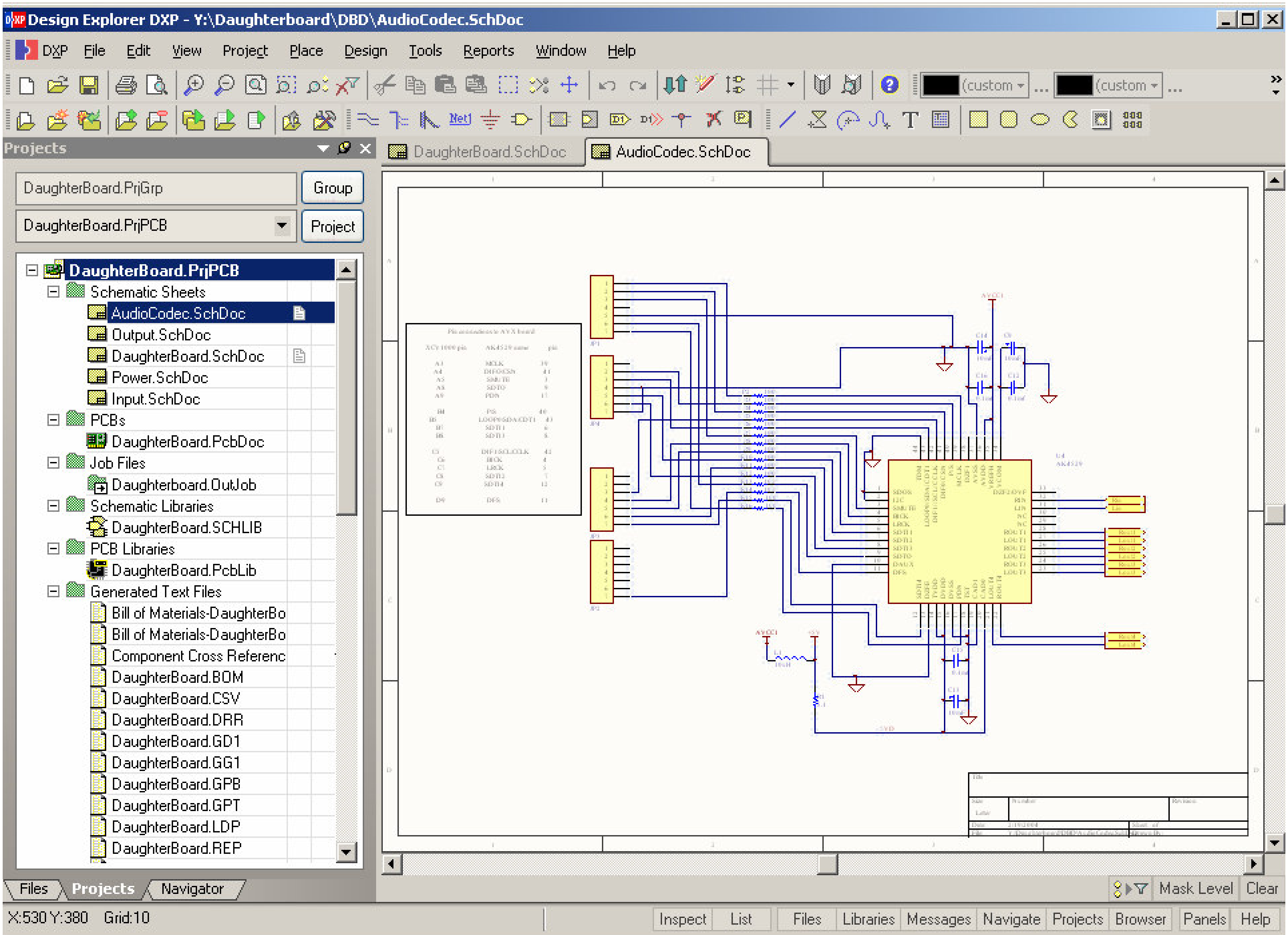


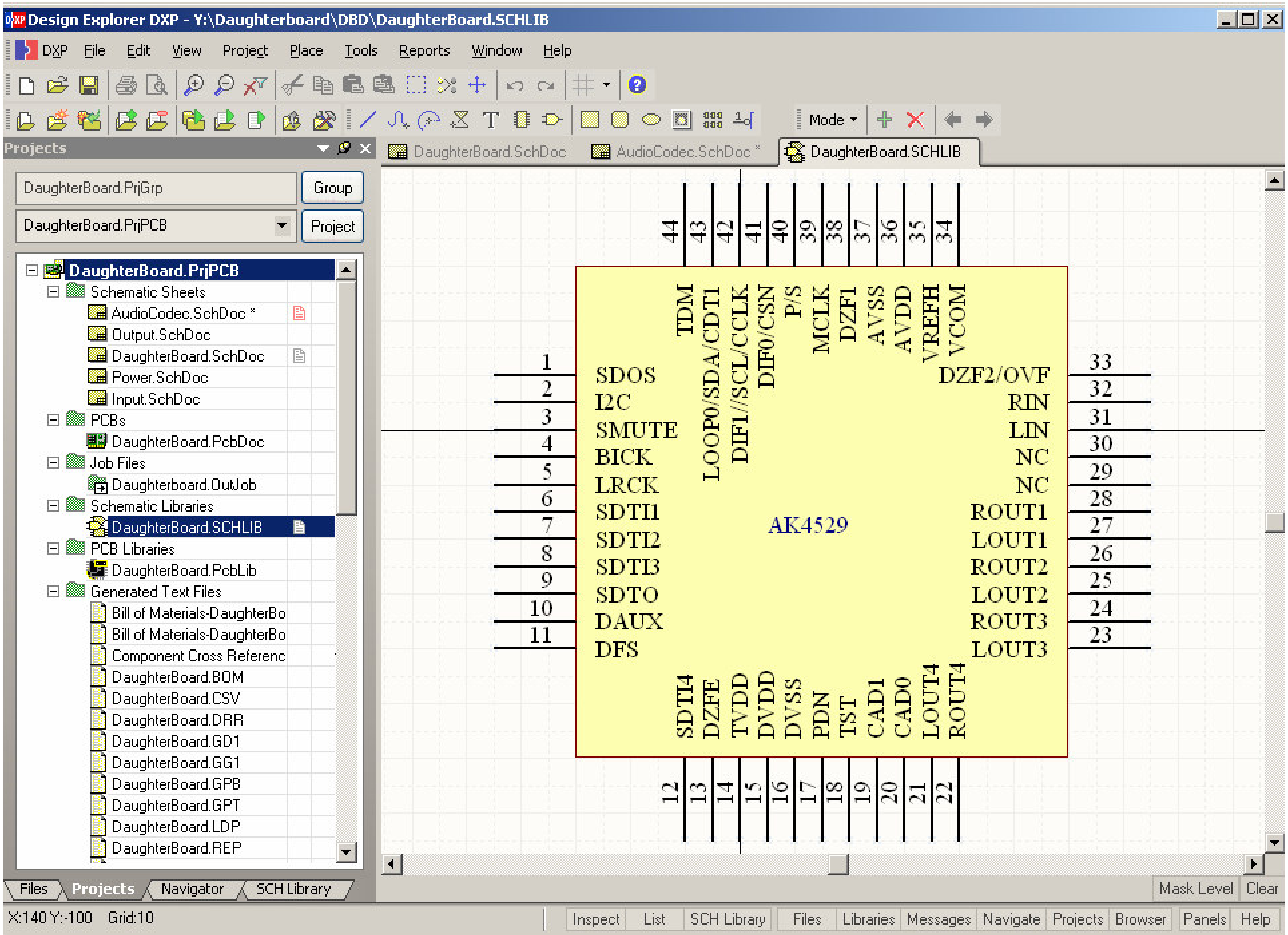
Protel

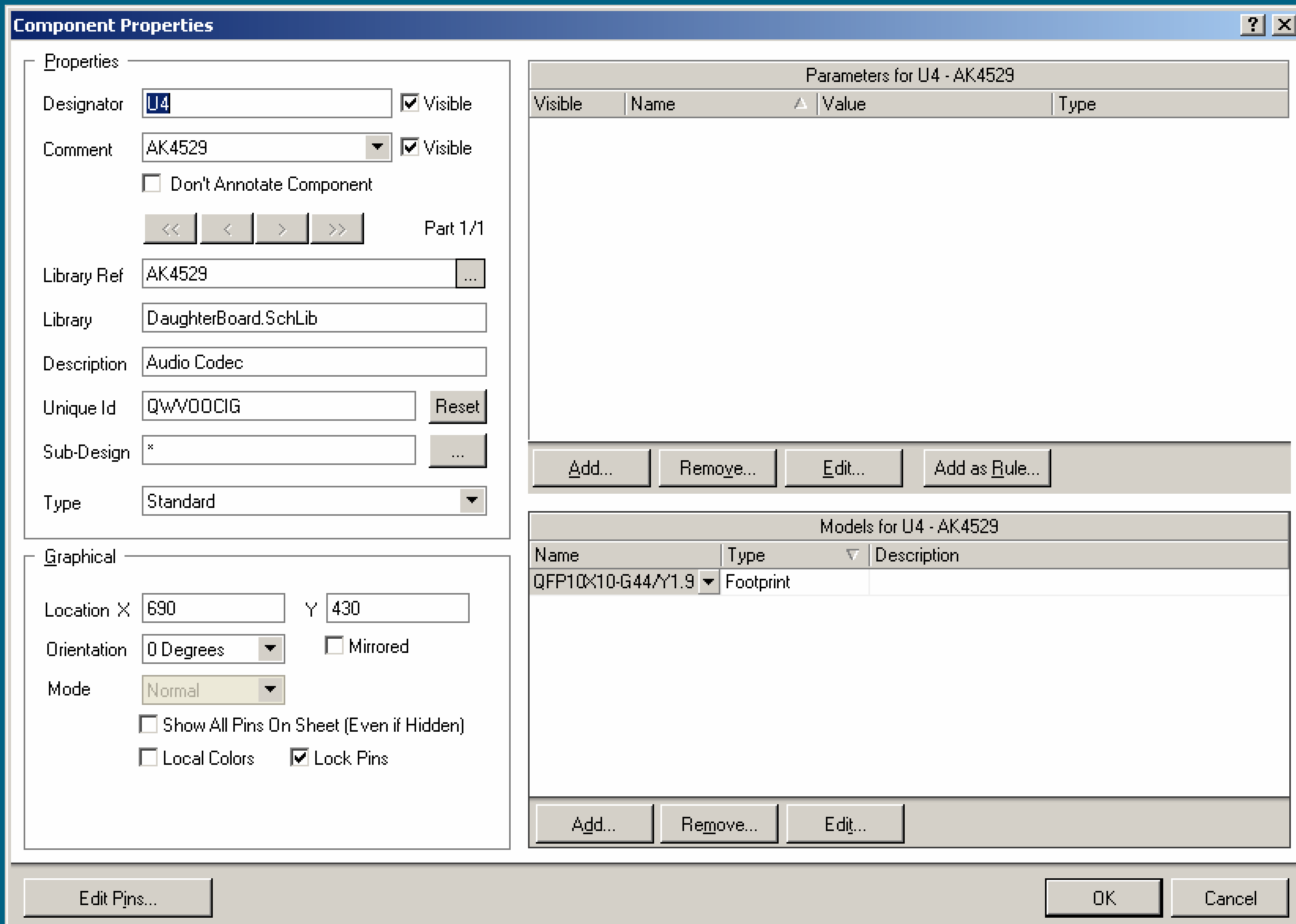
PCB Layout System

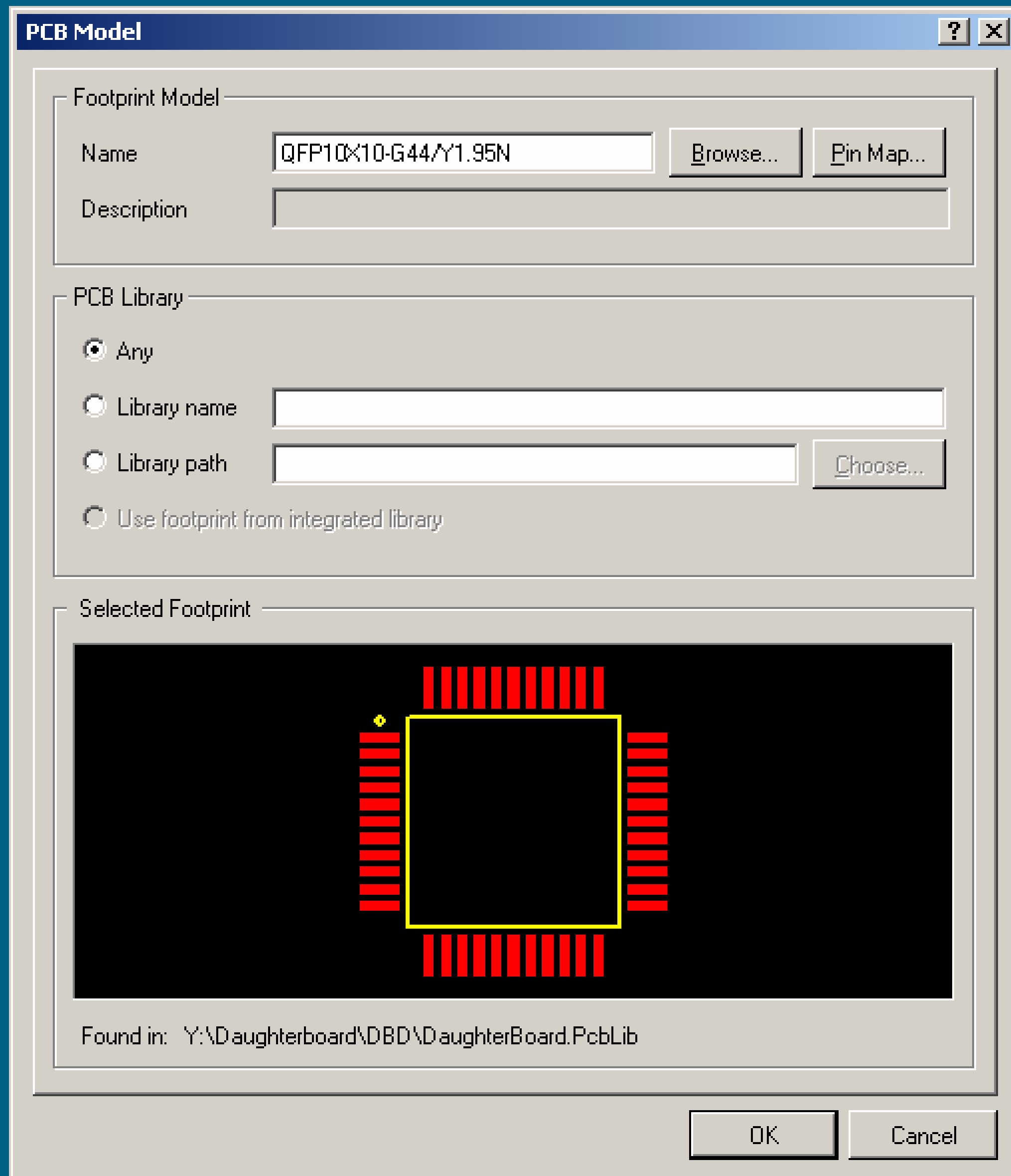

Design Explorer DXP
DXP File View Project Window Help
Projects
DaughterBoard.PrjGrp
Group
DaughterBoard.PrjPCB
Project
DaughterBoard.PrjPCB
Schematic Sheets
AudioCodec.SchDoc
Output.SchDoc
DaughterBoard.SchDoc
Power.SchDoc
Input.SchDoc
PCBs
DaughterBoard.PcbDoc
Job Files
Daughterboard.OutJob
Schematic Libraries
DaughterBoard.SCHLIB
PCB Libraries
DaughterBoard.PcbLib
Generated Text Files
Bill of Materials-DaughterBo
Bill of Materials-DaughterBo
Component Cross Referenc
DaughterBoard.BOM
DaughterBoard.CSV
DaughterBoard.DRR
DaughterBoard.GD1
DaughterBoard.GG1
DaughterBoard.GPB
DaughterBoard.GPT
DaughterBoard.LDP
DaughterBoard.REP
DaughterBoard.RUL
DaughterBoard.TXT
Pick Place for DaughterBoa
components
data
Files Projects Navigator
Pick a task
Create a new Board Level Design Project
Display System Information
Create a new FPGA Design Project
Customize Resources
Create a new Integrated Library Package
Configure Licenses
or open a project or document
Open a project or document
Most recent project - DaughterBoard.PrjPCB
Most recent document - Mixer_Routed.PCBDOC
or get help
DXP Online help
DXP Help Advisor
DXP Learning Guides
DXP Knowledge Base
Files Libraries Messages Navigate Projects Browser Panels Help

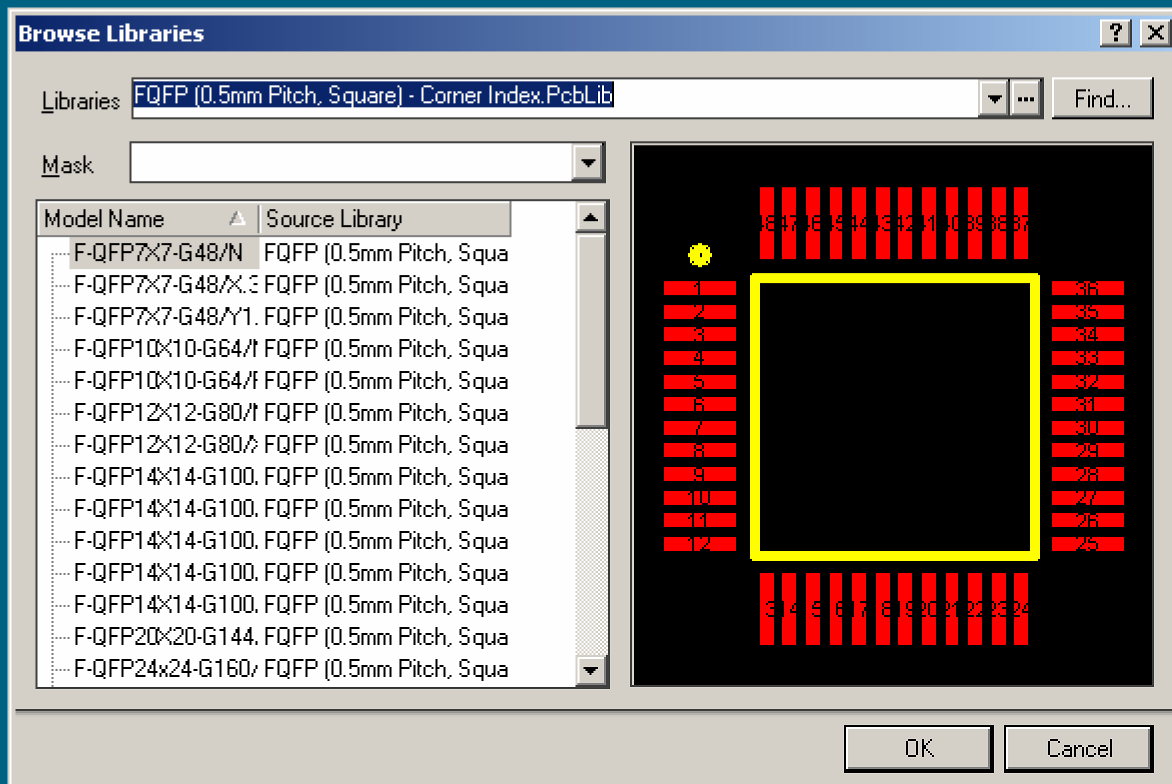

Design Explorer DXP - Y:\Daughterboard\DBD\DaughterBoard.SchDoc
DXP File Edit View Project Place Design Tools Reports Window Help
Projects
DaughterBoard.PrjGrp
Group
DaughterBoard.PrjPCB
Project
DaughterBoard.PrjPCB
Schematic Sheets
AudioCodec.SchDoc
Output.SchDoc
DaughterBoard.SchDoc
Power.SchDoc
Input.SchDoc
PCBs
DaughterBoard.PcbDoc
Job Files
Daughterboard.OutJob
Schematic Libraries
DaughterBoard.SCHLIB
PCB Libraries
DaughterBoard.PcbLib
Generated Text Files
Bill of Materials-DaughterBo
Bill of Materials-DaughterBo
Component Cross Referenc
DaughterBoard.BOM
DaughterBoard.CSV
DaughterBoard.DRR
DaughterBoard.GD1
DaughterBoard.GG1
DaughterBoard.GPB
DaughterBoard.GPT
DaughterBoard.LDP
DaughterBoard.REP
Files Projects Navigator
DaughterBoard.SchDoc
Mask Level Clear
X:480 Y:740 Grid:10
Inspect List Files Libraries Messages Navigate Projects Browser Panels Help

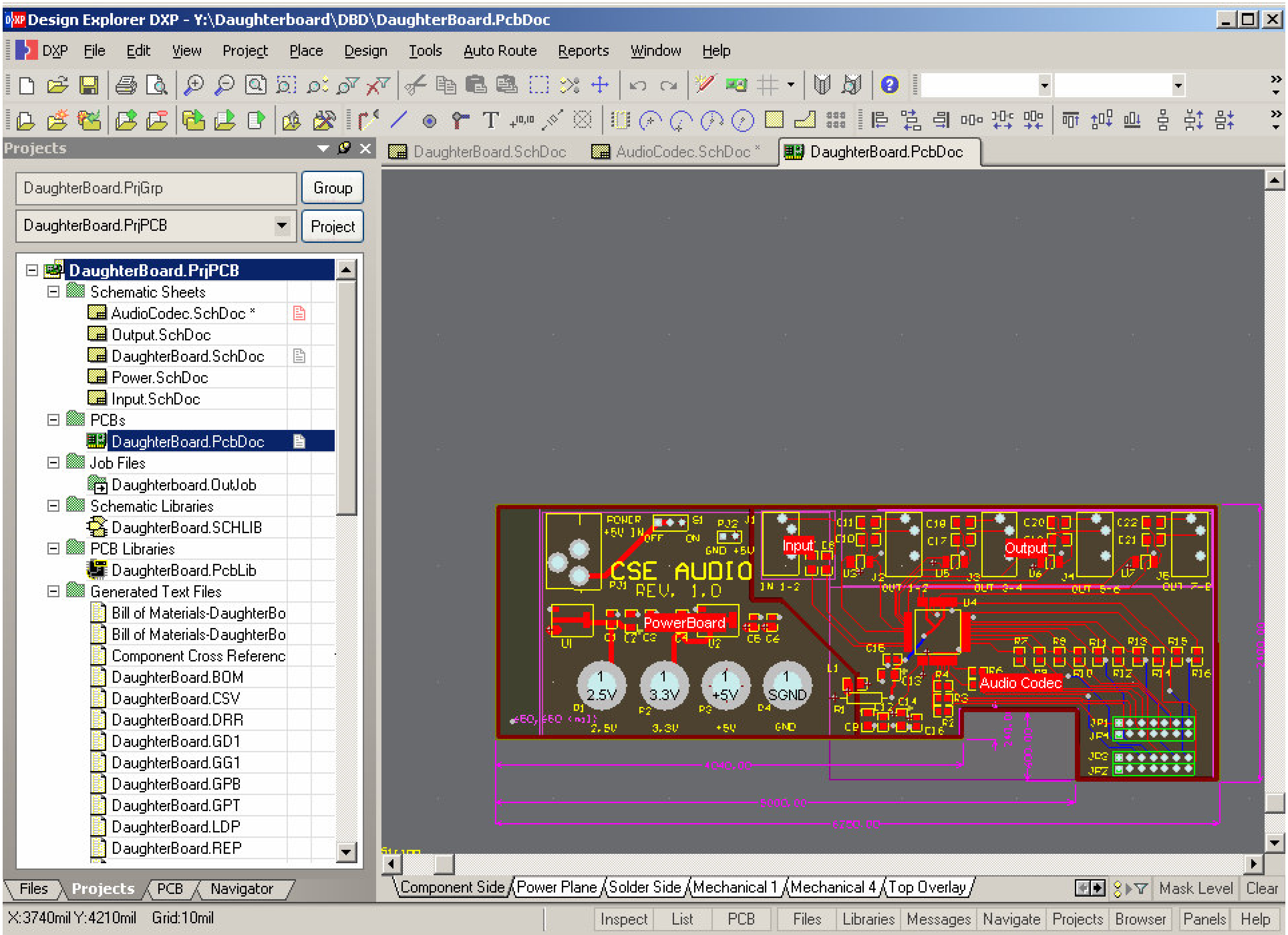

Design Explorer DXP - Y:\Daughterboard\DBD\AudioCodec.SchDoc
DXP File Edit View Project Place Design Tools Reports Window Help
(custom
Projects
DaughterBoard.SchDoc
AudioCodec.SchDoc
DaughterBoard.PrjGrp
Group
DaughterBoard.PrjPCB
Project
DaughterBoard.PrjPCB
Schematic Sheets
AudioCodec.SchDoc
Output.SchDoc
DaughterBoard.SchDoc
Power.SchDoc
Input.SchDoc
PCBs
DaughterBoard.PcbDoc
Job Files
Daughterboard.OutJob
Schematic Libraries
DaughterBoard.SCHLIB
PCB Libraries
DaughterBoard.PcbLib
Generated Text Files
Bill of Materials-DaughterBo
Bill of Materials-DaughterBo
Component Cross Referenc
DaughterBoard.BOM
DaughterBoard.CSV
DaughterBoard.DRR
DaughterBoard.GD1
DaughterBoard.GG1
DaughterBoard.GPB
DaughterBoard.GPT
DaughterBoard.LDP
DaughterBoard.REP
Files
Projects
Navigator
Mask Level
Clear
X:530 Y:380 Grid:10
Inspect
List
Files
Libraries
Messages
Navigate
Projects
Browser
Panels
Help


Design Explorer DXP - Y:\Daughterboard\DBD\DaughterBoard.SCHLIB
DXP File Edit View Project Place Tools Reports Window Help
Mode
Projects
DaughterBoard.SchDoc
AudioCodec.SchDoc *
DaughterBoard.SCHLIB
DaughterBoard.PrjGrp
Group
DaughterBoard.PrjPCB
Project
DaughterBoard.PrjPCB
Schematic Sheets
AudioCodec.SchDoc *
Output.SchDoc
DaughterBoard.SchDoc
Power.SchDoc
Input.SchDoc
PCBs
DaughterBoard.PcbDoc
Job Files
Daughterboard.OutJob
Schematic Libraries
DaughterBoard.SCHLIB
PCB Libraries
DaughterBoard.PcbLib
Generated Text Files
Bill of Materials-DaughterBo
Bill of Materials-DaughterBo
Component Cross Referenc
DaughterBoard.BOM
DaughterBoard.CSV
DaughterBoard.DRR
DaughterBoard.GD1
DaughterBoard.GG1
DaughterBoard.GPB
DaughterBoard.GPT
DaughterBoard.LDP
DaughterBoard.REP
AK4529
1 SDOS
2 I2C
3 SMUTE
4 BICK
5 LRCK
6 SDTI1
7 SDTI2
8 SDTI3
9 SDTO
10 DAUX
11 DFS
12 SDTI4
13 DZFE
14 TVDD
15 DVDD
16 DVSS
17 PDN
18 TST
19 CAD1
20 CAD0
21 LOUT4
22 ROUT4
23 LOUT3
24 ROUT3
25 LOUT2
26 ROUT2
27 LOUT1
28 ROUT1
29 NC
30 NC
31 LIN
32 RIN
33 DZF2/OVF
34 VCOM
35 VREFH
36 AVDD
37 AVSS
38 DZF1
39 MCLK
40 P/S
41 DIF0/CSN
42 DIF1/SCL/CCLK
43 LOOP0/SDA/CDTI
44 TDM
Files Projects Navigator SCH Library
Mask Level Clear
X:140 Y:-100 Grid:10
Inspect List SCH Library Files Libraries Messages Navigate Projects Browser Panels Help

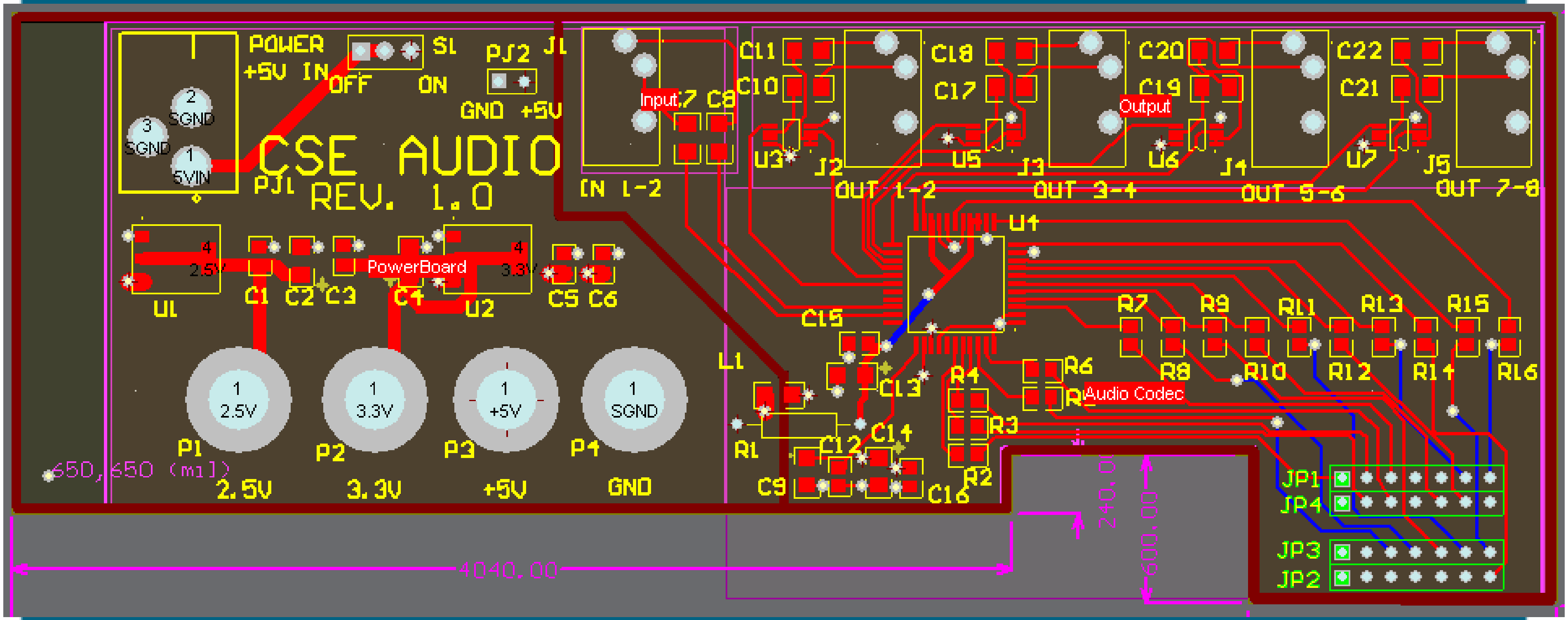

Component Properties
Properties
Designator
U4
Visible
Comment
AK4529
Visible
Don't Annotate Component
Part 1/1
Library Ref
AK4529
Library
DaughterBoard.SchLib
Description
Audio Codec
Unique Id
QWVOOCIG
Reset
Sub-Design
Type
Standard
Graphical
Location X
690
Y
430
Orientation
0 Degrees
Mirrored
Mode
Normal
Show All Pins On Sheet (Even if Hidden)
Local Colors
Lock Pins
Parameters for U4 - AK4529
Visible
Name
Value
Type
Add...
Remove...
Edit...
Add as Rule...
Models for U4 - AK4529
Name
Type
Description
QFP10X10-G44/Y1.9
Footprint
Add...
Remove...
Edit...
Edit Pins...
OK
Cancel


PCB Model
Footprint Model
Name
QFP10X10-G44/Y1.95N
Browse...
Pin Map...
Description
PCB Library
Any
Library name
Library path
Choose...
Use footprint from integrated library
Selected Footprint
Found in: Y:\Daughterboard\DBD\DaughterBoard.PcbLib
OK
Cancel


Browse Libraries
Libraries
FQFP (0.5mm Pitch, Square) - Corner Index.PcbLib
Find...
Mask
Model Name
Source Library
F-QFP7X7-G48/N FQFP (0.5mm Pitch, Squa
F-QFP7X7-G48/X.3 FQFP (0.5mm Pitch, Squa
F-QFP7X7-G48/Y1. FQFP (0.5mm Pitch, Squa
F-QFP10X10-G64/I FQFP (0.5mm Pitch, Squa
F-QFP10X10-G64/I FQFP (0.5mm Pitch, Squa
F-QFP12X12-G80/I FQFP (0.5mm Pitch, Squa
F-QFP12X12-G80/ FQFP (0.5mm Pitch, Squa
F-QFP14X14-G100. FQFP (0.5mm Pitch, Squa
F-QFP14X14-G100. FQFP (0.5mm Pitch, Squa
F-QFP14X14-G100. FQFP (0.5mm Pitch, Squa
F-QFP14X14-G100. FQFP (0.5mm Pitch, Squa
F-QFP14X14-G100. FQFP (0.5mm Pitch, Squa
F-QFP20X20-G144. FQFP (0.5mm Pitch, Squa
F-QFP24x24-G160/ FQFP (0.5mm Pitch, Squa
OK
Cancel



Design Explorer DXP - Y:\Daughterboard\DBD\DaughterBoard.PcbDoc
DXP File Edit View Project Place Design Tools Auto Route Reports Window Help
Projects
DaughterBoard.PrjGrp
Group
DaughterBoard.PrjPCB
Project
DaughterBoard.PrjPCB
Schematic Sheets
AudioCodec.SchDoc *
Output.SchDoc
DaughterBoard.SchDoc
Power.SchDoc
Input.SchDoc
PCBs
DaughterBoard.PcbDoc
Job Files
Daughterboard.OutJob
Schematic Libraries
DaughterBoard.SCHLIB
PCB Libraries
DaughterBoard.PcbLib
Generated Text Files
Bill of Materials-DaughterBo
Bill of Materials-DaughterBo
Component Cross Referenc
DaughterBoard.BOM
DaughterBoard.CSV
DaughterBoard.DRR
DaughterBoard.GD1
DaughterBoard.GG1
DaughterBoard.GPB
DaughterBoard.GPT
DaughterBoard.LDP
DaughterBoard.REP
DaughterBoard.SchDoc
AudioCodec.SchDoc *
DaughterBoard.PcbDoc
CSE AUDIO
REV. 1.0
PowerBoard
Input
Output
Audio Codec
2.5V
3.3V
+5V
SGND
Files Projects PCB Navigator
Component Side Power Plane Solder Side Mechanical 1 Mechanical 4 Top Overlay
Mask Level Clear
X:3740mil Y:4210mil Grid:10mil
Inspect List PCB Files Libraries Messages Navigate Projects Browser Panels Help


POWER
+5V IN
OFF
ON
S1
PJ2
J1
GND +5V
CSE AUDIO
REV. 1.0
PJ1
IN 1-2
Input
OUT 1-2
OUT 3-4
OUT 5-6
OUT 7-8
Output
PowerBoard
Audio Codec
U1
U2
U3
U4
U5
U6
U7
P1
P2
P3
P4
2.5V
3.3V
+5V
GND
JP1
JP2
JP3
JP4
4040.00