Designer's Checklist

Circuit Designer Requirements

Initial Planning

- ❒ Gather required information for part list, required component locations, and mechanical locations and requirements.
- ❒ Determine if all components are available in existing libraries. If not, use a component creation checklist.
- ❒ Select design template.
- ❒ Save file by part number.
- ❒ Enter design information.
- ❒ Open/load necessary libraries.
- ❒ Place components and wire together.
- ❒ Note all current, voltage, high frequency, noise and circuits.
- ❒ Add a note like the following for the most commonly used trace and space in the board designs: *"Unless otherwise specified, all circuits are less than .25A and 30V" (this works for 6/6; .006" trace & .006" space).*
- ❒ Place power-pin table.
- ❒ Place "last used" and "unused pin/gate" table.
- ❒ Highlight power nets and check each sheet for connectivity.
- ❒ Check for design rules, such as single node net; no node net; unconnected pins; unconnected wires; or other.
- ❒ Generate BOM and compare against part list.
- ❒ Add necessary notes.
- ❒ Add sheet numbers.
- ❒ Print and check schematic visually.
- ❒ Align/modify location, format and styles.
- ❒ Place all nets into classes.
- ❒ Generate netlist.
- ❒ Archive libraries.
- ❒ Check for other concerns:
    - ❒ *Are all IC inputs terminated as required?*
    - ❒ *Do IC/components have necessary filter caps?*
    - ❒ *Are main circuit and branch circuits clearly identified?*

Printed Circuit Board Design Checklist

(This topic covered in Chapter 5)

Define Constraints

- ❒ Define board dimensions.
- ❒ Define top and bottom board clearance.
- ❒ Note dimensions of cutouts slots and unusable areas.
- ❒ Define the board thickness.
- ❒ Define edge clearance areas.
- ❒ Define all slots and cutouts.
- ❒ Define assembly requirement such as keying information.
- ❒ Mark predefined component locations including hardware, connectors, lights/LEDs, and switches.
- ❒ Place polygon on mask layer for area that requires no mask.
- ❒ Place keep-out (all layers or per layer) on area that require clearance from/for traces, vias, pads, and hardware clearance (by hardware-to-hole tolerance, hardware movement area, or hole tolerance).
- ❒ Define requirements: IPC, Mil-spec, etc.
- ❒ Determine assembly type for production, such as manual, automatic, or manual prototype-to-automatic.
- ❒ Determine servicing type:
    - ❒ *No service/troubleshooting (throw away board)*
    - ❒ *Low service (inexpensive components on the board, or easily swappable application, low serviceable location)*
    - ❒ *Highly serviceable (expensive components on the board or difficult to swap application, highly serviceable location)*
    - ❒ *Determine technology limitations and target technology.*

Begin Design

- ❒ Open new file or load appropriate template.
- ❒ Check for standards in pads, vias, or text styles.
- ❒ Draw board border using .040” line on center.
- ❒ Draw all slots/cutouts in board using .040” line on center.
- ❒ Enter design information.
- ❒ Load libraries/archived library.

- ❒ Load netlist.
- ❒ Generate BOM and compare against parts list. (This is to include mechanical components not in the schematic.)
- ❒ Place parts with a predefined location where necessary.
- ❒ Define classes/nets with trace width, clearance (space), and hole clearance
- ❒ Define other design attributes or design rules.
- ❒ If applicable, define class trace and space by layer.
- ❒ Configure design/job for:
    - ❒ *Overall design rules*
    - ❒ *Mask swell (global)*
    - ❒ *Paste swell (global)*
    - ❒ *Plane swell (global)*
    - ❒ *Pad swell (global)*
    - ❒ *Thermal divide (pad dia./4)*
    - ❒ *Thermal clearance*
- ❒ Define assembly direction (especially auto assembly).
- ❒ Define/determine component direction.
- ❒ Define areas by type.
- ❒ Define layers including:
    - ❒ *Number*
    - ❒ *Symmetry (signal plane signal, etc.)*
    - ❒ *Layer direction*
    - ❒ *Layer type (strictly power, digital, etc.)*
    - ❒ *Split planes*
- ❒ Copy board, cutout, slot outlines to all plane layer providing copper to edge clearances.
- ❒ Add text to plane layer as to net name (GND, +5V, etc.).
- ❒ Calculate board thickness and determine material availability. Attempt to use predefined or previously used combinations. Or, after design completion save successful stack-up combinations.
- ❒ Add tooling holes, if appropriate.
- ❒ Add datums to all layers or overlay layer. This helps not only to verify alignment after completion but for manufacturing alignment.
- ❒ Board part number in copper on bottom side
- ❒ Board revision (in copper or manually marking)

- ❒ Layer number (each layer, numbered by layer number, each offset)
- ❒ Assembly number (on silkscreen, top-side)
- ❒ Assembly revision (leave blank area for manual marking)

Initial Checks

- ❒ Check power pins are connected correctly on 1 of each type of part.
- ❒ Check plated-mounting holes are grounded when required.
- ❒ Complete placement location and prepare for routing.

Manual Routing

- ❒ Route the following types of nets first:
    - ❒ *Most difficult*
    - ❒ *Most complex*
    - ❒ *Tight fitting nets first*
    - ❒ *Very high current (primarily external)*
    - ❒ *Very high voltage (primarily internal)*
    - ❒ *Sensitive*
    - ❒ *Noisy*
- ❒ Separate analog and digital.
- ❒ Route busses.

Auto Routing

- ❒ Manually route those items shown in “manual routing” first, if necessary.
- ❒ Define attributes that are common only to the auto router.
- ❒ Define/select “Routine,” “Do” file, “Route” file, or “Strategy” file.
- ❒ After route completion:
    - ❒ *Manually clean up paths.*
    - ❒ *Miter right angle corners.*
    - ❒ *Run DRC/design rules to ensure clearances are met.*
    - ❒ *Check annular ring.*
    - ❒ *Change gates or parts.*

Additional Markings

- ❒ ESD symbol
- ❒ High voltage warning
- ❒ Run DRC/design rules to ensure clearances are met.
- ❒ Relocate reference designators to their correct position/location/orientation.
- ❒ Relabel/renumber reference designator.

Creating a Manufacturing/Fabrication Drawing

- ❒ Copy border(s) to drawing layer or include border layer.
- ❒ Dimension the board in X and Y dimensions.
- ❒ Hole to edge dimensions (This is used for registration verification for Gerber/drill loading)
- ❒ Dimension and tolerance of any cutouts under +/- .005" tolerance.
- ❒ Board lay-up including the following:
    - ❒ *Layer number*
    - ❒ *Layer type*
    - ❒ *Layer thickness*
    - ❒ *Layer tolerance*
    - ❒ *Copper layer type*
    - ❒ *Min trace width spacing per layer (special cases only)*
    - ❒ *Overall board thickness*
    - ❒ *Overall board tolerance (Conventional +/-10%)*
- ❒ Drill legend, including:
    - ❒ *Finished hole size*
    - ❒ *Hole type (Plated or non-plated)*
    - ❒ *Hole tolerance (Holes under .080" +/-.003". Holes over .080" +/-.005". Changes per technology)*
    - ❒ *Symbol (Correlates with fabrication drawing or Gerber export)*
- ❒ Load or define fabrication notes including:
    - ❒ *Guidelines or specifications to follow unless otherwise noted (PC class quality) and type SS/DS or ML)*
    - ❒ *Material used (Core and pre-preg)*
- ❒ Is copper thickness specified per table?
- ❒ Min trace width and tolerance (+/-.003" general .001" tight)
- ❒ Min clearance and tolerance (+/-.003" general .001" tight)
- ❒ Plating per same table (more plating, more plating in hole, increased MFG AR)
- ❒ Hole plating minimum of .0002 (usually external plating)
- ❒ Finish type: HASL or tin lead (check for availability)
- ❒ Hole to pad registration (no breakout allowed)
- ❒ Layers to layer 1 registration (+/-.002")

- ❒ Overall scale tolerance (+/-.002 per inch. +/-.005” overall).
- ❒ Board size tolerance (+/-.005”)
- ❒ Slot tolerance (+/-.003” to −.005”)
- ❒ Beveling (if required)
- ❒ Electrical test and receipt of official test results. If required by P.O. or prototype only. Continuity of less than 5 Ohms per inch. Test at 100 V.
- ❒ One or more of the following manufacturing markings (usually placed on the bottom side):
    - ❒ *Cage code (normally used by military contractors)*
    - ❒ *Company logo (for identification if additional parts need to be ordered later in time)*
    - ❒ *Date code (for board history)*
    - ❒ *Lot code (for troubleshooting)*
    - ❒ *Electrical test verification marking*
- ❒ Twist & bow value (.010” def., .007” tight)
- ❒ Coupon or x-ray inspection for hole wall quality (one of the most important quality aspects of a board)
- ❒ Other

Documentation

- ❒ Sheet/numbers of sheets
- ❒ Load or add information block specifying the following (this information may stay with the board until it is removed from the panel):
    - ❒ *Company name*
    - ❒ *Company phone*
    - ❒ *Layer name*
    - ❒ *Layer number*
    - ❒ *Part number*
    - ❒ *Revision*
    - ❒ *Sheet of sheets*

Application Company Specific Information

- ❒ Add sheet revision block on first page (fabrication drawing).
- ❒ Add sheet revision section (border information).
- ❒ Update design information such as the following:
    - ❒ *Date (update every time this file is finished, changed, or modified)*
    - ❒ *Designed by (designer name)*

- ❒ *Engineer (electrical engineer or the schematic's entry person)*
- ❒ *Checked by (QC, final, or engineer's name)*

❒ Add sheet revision block on first page. (fabrication drawing)

Check Plots (not required)

❒ Print each layer w/o boarder to scale.

❒ Inspect for the following:

- ❒ *Sheet layer numbers*
- ❒ *Datums*
- ❒ *ESD symbol*
- ❒ *HV note*
- ❒ *Tooling*
- ❒ *Pin 1 identification*
- ❒ *Mounting hole locations*
- ❒ *Board size and clearance*
- ❒ *Mechanical support*
- ❒ *Hardware clearance*
- ❒ *Stack-up thickness*

Approval

❒ Get PCB approval from engineer.

❒ Implement any redlines.

❒ Generate netlist from schematic again.

❒ Run DRC again and run compare netlist.

Output

❒ Set up Gerber output files or set up database export.

❒ Export the following (in 274-X):

- ❒ *All layers separately*
- ❒ *Required silk screen layers*
- ❒ *Top and bottom solder mask separately*
- ❒ *Fabrication drawing with symbols*
- ❒ *Drill file (in ASCII format, leading suppression)*

❒ Load Gerbers in a CAM/CAD viewer and inspect for consistency with original design.

File Archive

- ❒ May be done after receipt of board or when test is complete.
- ❒ Place files in restricted area and change file properties as Read-only.
- ❒ If changes need to be made, change revision and start a new file using revision descriptor in the part number.

Incoming Board Inspection

(This topic covered in Chapter 7)

- ❒ Check initial look of board. Note cleanliness and appearance.
- ❒ Check mask, including:
    - ❒ *Specified color*
    - ❒ *Specified thickness*
    - ❒ *Quality*
    - ❒ *Blemished*
    - ❒ *Pitting*
- ❒ Is plating adequate?
- ❒ Are holes centered in pad (annular ring and alignment)?
- ❒ Are hole sizes correct?
- ❒ Are layers registered?
- ❒ Adhesion test
- ❒ Does appearance match artwork?
- ❒ Does silkscreen match artwork?
- ❒ Are overall board dimensions correct?
- ❒ Is board warped?
- ❒ Is any copper showing on board edges?
- ❒ Trace width in tolerance
- ❒ Check electrically the following items:
    - ❒ *Several plated holes front to back (less than 2 Ohms)*
    - ❒ *Longest trace from one end to the other (less than 5 Ohms per inch, or specified value)*
    - ❒ *Resistance between planes of different nets for shorting (should be open)*
    - ❒ *Test pads closest place for continuity (known opens should read open)*

Printed Circuit Board Assembly Checklist

Create an Assembly Drawing

(This topic covered in Chapter 8)

- ❒ Load title block on top assembly layer.
- ❒ Add page # to each sheet.
- ❒ Enter design info on assembly fields or designated areas.
- ❒ Load basic notes according to board type.
- ❒ Load silk Gerber, remove all exterior to the board and copy to top assembly.
- ❒ Draw side/bottom view of board (including parts and screws, etc.).
- ❒ Using parts list, add find number leader with quantity.
- ❒ Add notes as required and triangle find numbers as needed.
- ❒ Place additional tables for any wire(s) used. Always add note and triangle find number with basic note.
- ❒ Check for mounting hardware on all components and sub-assemblies.
- ❒ Check soldering, shrink (all soldered terminals), glue locations, required fixture, mounting brackets, and card guides.

Conformal Coating only

- ❒ Place phantom lines.
- ❒ Place dimensions for conformal coat clearance with note find number.
- ❒ Place serial number box block next to the serial number silkscreen.
- ❒ Add appropriate additional notes and place find numbers
- ❒ Place appropriate distribution statements and proprietary notes
- ❒ Place revision sheet information on all layers.
- ❒ Place revision status block on assembly drawing, top sheet.
- ❒ Get assembly drawing(s) approval.
- ❒ Compress Gerbers into *drawing number*.ext.

Quick Tables

Table 1: Technology Table for Conventional Technology

Type	Conventional	Note
	Etching	
Etch back	.0014	Per 1oz copper (.0007 per copper side)
Trace Min (1/2oz starting)	.006”	Per starting copper
Trace Min (1oz starting)	.007”	
Trace Min (2oz starting)	.008”	
Trace Min (3oz starting)	.010”	
Space Min (1/2oz starting)	.006”	Per starting copper
Space Min (1oz starting)	.007”	
Space Min (2oz starting)	.008”	
Space Min (3oz starting)	.010”	
Trace Tol. (+/- for 1/2oz)	.002”	+/- value (Plating has no effect)
Trace Tol. (+/- for 1oz)	.002”	
Trace Tol. (+/- for 2oz)	.003”	
Trace Tol. (+/- for 3oz)	.003”	
	Copper Plating	
Min. External Copper (oz) Starting	½ OZ CU.	
Min. External Copper (oz) Finished	1 1/2oz	
Max. External Copper (oz)	4 OZ	
Min. Internal Copper (oz)	½ OZ CU.	
Max Internal Copper (oz)	3 OZ CU.	
Starting/Plating oz for 1 1/2oz Finished	½/1	Starting/plating (oz) 1/2oz = .0007”
Starting/Plating oz for 2 1/2oz Finished	1/1	Starting/plating (oz) 1/2oz = .0007”
Starting/Plating oz for 3 1/2oz Finished	2/1	Starting/plating (oz) 1/2oz = .0007”
Starting/Plating oz for 4 1/2oz Finished	3/1	Starting/plating (oz) 1/2oz = .0007”
Hole wall Plating/Solder	.012”/.012” (+/-.002”ea)	For solder finished (Plating inch/Solder inch)

Type	Conventional	Note
	Drill	
Min. Drill (Drilled Hole)	.012	Finished drill = Min drill – "drill over" (*unless plating thickness is increased*)
Min Aspect Ratio	5:1	.062" Board = .012" drill min
Drill tol. PLTH (<.080)	+/-.003	
Drill tol. PLTH (>.080)	+/-.004	
Drill tol. NPTH (<.080)	+/-.002	
Drill tol. NPTH (>.080)	+/-.003	
Trace to Hole (ML)	.009" Min	Clearance (Not including electrical clearance)
Trace to Hole (SS/DS)	.006" Min	Clearance (Not including electrical clearance)
Drill to pad (ML)	.009"	From Center (Image + Drill + Lay-up registration)
Drill to pad (SS/DS)	.006"	From Center (Image + Drill)
Board edge	.010" min	Clearance for route
Image Registration Error	.003"	Max allowable
Drill Registration Error	.003"	Max allowable
Lay-up Registration Error	.003"	Max allowable
ML total	.009	ML includes image, drill & lay-up
SS/DS total	.006	SS/DS includes Image, drill only.
	Material	
Min. Pre-preg Material thick	.002	Thickness of a single sheet of Pre-preg
Board Max.	.125	Thickness (FR4)
Board Min. (DS)	.008" + 1/2oz + 1/2oz copper .0094"	Excluding Plating
Min. ML	.020"	Excluding Plating (4layer)
Min. Thick ML Tolerance	+/-10% or total material tol.	%
Min. Material tolerance	+/-10%	Std Fr4
	Mask	
LPI Min clear	.002"	
LPI Swell AR	.004"	
LPI Reg.	+/-.005"	
LPI Avg. thick	.0015	
LPI Min thick	.0008	
LPI color	GREEN, BLUE, RED, BLACK	

Type	Conventional	Note
	Silk Screen	
SS Reg.	+/-.006”	
SS Height	.080”	
SS Thick	.008”	Line width
SS color	WHITE, YELLOW, RED, BLACK	
	Surface Mount	
SM Pitch	.015	Pad center-center (Text purposes only)
SM PAD	.008	Same as Trace width
	Miscellaneous	
Bow & Twist	.010 Per inch	
Board Max.	18 X 15	X” / Y”
# of layers	12	MAXIMUM
Imp Tolerance	N/A	+/- % (If applicable)

Note: AR=Annular Ring, SS= Silk Screen, MSK= Mask

Table 2: Width added for etch factor (Conventional)

Starting Oz.	Add…
.5	.0007”[.0118]
1	.0014”[.0356]
2	.0028”[.0711]
3	.0042”[.1067]

Table 3: Quick table of finished aspect ratios

Board Thickness	For 5:1 (Conventional) Min drill .018”	For 8:1 (Advanced) Min drill .012”	For 10:1 (Leading Edge) Min drill .008”
.010	.002	.001	.001
.020	.004	.002	.002
.030	.006	.003	.003
.040	.008	.005	.004
.050	.010	.006	.005
.060	.012	.007	.006
.070	.014	.008	.007
.080	.016	.010	.008
.090	.018	.011	.009
.100	.020	.013	.010
.125	.025	.016	.012
.150	.030	.019	.015
.175	.035	.022	.017

Note: Multiply these values by 25.4 for mm; Grey areas are below minimum drill size

Table 4: Expensive board

Possibility of replacement	Board space requirements	Pad Size	
High	Low	Ideal	2 X hole diameter
High	Medium	Ideal	2 X hole diameter
High	High	Nominal	1.75 X hole diameter
Medium	Low	Ideal	2 X hole diameter
Medium	Medium	Ideal	2 X hole diameter
Medium	High	Nominal	1.75 X hole diameter
Low	Low	Ideal	2 X hole diameter
Low	Medium	Nominal	1.75 X hole diameter
Low	High	Minimum	1.5 X hole diameter

Table 5: Moderate board

Possibility of replacement	Board space requirements	Pad Size	
High	Low	Ideal	2 X hole diameter
High	Medium	Ideal	2 X hole diameter
High	High	Nominal	1.75 X hole diameter
Medium	Low	Ideal	2 X hole diameter
Medium	Medium	Nominal	1.75 X hole diameter
Medium	High	Minimum	1.5 X hole diameter
Low	Low	Nominal	1.75 X hole diameter
Low	Medium	Nominal	1.75 X hole diameter
Low	High	Minimum	1.5 X hole diameter

Table 6: Inexpensive board

Possibility of replacement	Board space requirements	Pad Size	
High	Low	Ideal	2 X hole diameter
High	Medium	Nominal	1.75 X hole diameter
High	High	Minimum	1.5 X hole diameter
Medium	Low	Nominal	1.75 X hole diameter
Medium	Medium	Nominal	1.75 X hole diameter
Medium	High	Minimum	1.5 X hole diameter
Low	Low	Nominal	1.75 X hole diameter
Low	Medium	Minimum	1.5 X hole diameter
Low	High	Minimum	1.5 X hole diameter

Table 7: Manual/Auto assembly decision matrix

Component types	Spacing/ Density	Production Quantities	Auto assembly	Manual assembly
Thru-Hole	High	High	Yes	No
	High	Medium	Yes	No
	High	Low	Yes	No
	Medium	High	Yes	No
	Medium	Medium	Yes	No
	Medium	Low	No	Yes
	Low	High	Yes	No
	Low	Medium	No	Yes
	Low	Low	No	Yes
SMT	High	High	Yes	No
	High	Medium	Yes	No
	High	Low	Yes	No
	Medium	High	Yes	No
	Medium	Medium	Yes	No
	Medium	Low	Yes	No
	Low	High	Yes	No
	Low	Medium	Yes	No
	Low	Low	No	Yes
Mixed	High	High	Yes	No
	High	Medium	Yes	No
	High	Low	No	Yes
	Medium	High	Yes	No
	Medium	Medium	Yes	No
	Medium	Low	No	Yes
	Low	High	Yes	No
	Low	Medium	No	Yes
	Low	Low	No	Yes

Table 8: Schematic style standards

	Yes	No
Sheet size A used?		X
Sheet size B used?	X	
Sheet size C used?		X
Sheet size D used?	X	
IPC Drawing Standards used?	X	
IEEE standards used?	X	
ANSI drawing standards used?	X	
Sheets in Columnar format?	X	
Sheet in generic end-to-end format?	X	
Sheet in continuous sheet format?	X	
Sheet zones shown?	X	
Continuous sheet zones shown?		X
Sheet connectors Show Page & Zone?	X	
Sheet Connectors Show zone only		X
Sheet connectors show net name only		X
Connector block?	X	
Connectors Individual?		X
Connector shows block/grouping lines?		
Components shown as blocks?	X	
Components shown as DeMorgan?		
Components shown as IEEE?		
Components shown as gates?	X	
Filter caps shown in block with Component?		
Components Display Component Type?		
Component Pin names shown?		
Power pins Displayed?		
Power pins hidden? (Use PP table)	X	
All Net names displayed?	X	
Selective Net names Displayed?		
No net names Displayed?		

Table 9: Component letter standards

Letter	Component	Letter	Component
K	Relay	Z	Zener Diode
R	Resistor	W	Wire
J or P	Connector	X	Transformer
D	Diode	Q	Transistor
C	Capacitor	F	Fuse
L or H	Inductor or Coil	R	Potentiometer
L	Led or Lamp		
T or TP	Test point		
Symbol only	Ground (common)		
Symbol only	Ground (Chassis)		
Symbol only	Ground (Digital)		
Symbol only	Ground		

Table 10: Schematic control & text standards for primary drawing format

Drawing sizes	A&B	C&D
Component Pins	.200”[5.08]/ .100”[2.54]/ .025”[.635]	.400”[10.16]/ .200”[5.08]/ .100”[2.54]
Text Heights		
Reference Designator	.100”[2.54]	.200”[5.08]
Connector Reference Designator	.150”[3.81]	.250”[6.35]
Component Values	.075”[1.905]	.150”[3.81]
Notes	.100”[2.54]	.200”[5.08]

Note: These values are dependant upon the programs text measurements and may need adjusted to fit other formats.

Table 11: Quick reference guide limits per technology

Attribute	Conventional		Advance		Leading Edge	
	Use	Minimum	Use	Minimum	Use	Minimum
Min. Finished Drill	.014”[.3556]	.012”[.3048]	.012”[.3048]	.010”[.254]	.010”[.254]	.008”[.2332
Min plated hole for: (based on .005” plating)						
.031” board	.013” [.3302]		.007”[.1778]		.003”[.0762]	
.042” board	.013” [.3302]		.007”[.1778]		.003”[.0762]	
.080” board	.013” [.3302]		.007”[.1778]		.005”[.0762]	
.100” board	.013” [.3302]		.008”[.2332]		.008”[.2032]	
Clearances (Starting copper)						
Clearance (.5oz)	.008”[.2032]	.006”[.1524]	.006”[.1524]	.005”[.127]	.004”[.1016]	.003”[.0762]
Clearance (1oz)	.010”[.254]	.007”[.1778]	.006”[.1524]	.006”[.1524]	.006”[.1524]	.004”[.1016]
Clearance (2oz)	.012”[.3084]	.008”[.2032]	.010”[.254]	.007”[.1778]	.008”[.2032]	.005”[.0762]
Clearance (3oz)	.014”[.3556]	.010”[.254]	.012”[.3084]	.008”[.2032]	.010”[.254]	.008”[.2032]
Plane to edge	.020”[.508]	.008”[.2032]	.010”[.254]	.006”[.1524]	.008”[.2032]	.003”[.0762]
Hole clearance (SS/DS) +electrical clearance	.008”[.2032]	.006”[.1524]	.006”[.1524]	.004”[.1016]	.004”[.1016]	.002”[.0508]
Hole clearance (ML) +electrical clearanc	.010”	.009”	.008”[.2032]	.006”[.1524]	.005”	.003”
Lead clearance	.010”		.010”		.010”	
Width (.5oz)	.008”[.2032]	.006”[.1524]	.006”[.1524]	.005”	.004”	.003”
Width (1oz)	.010”	.007”	.008”	.006”[.1524]	.006”[.1524]	.004”
Width (2oz)	.012”	.008”[.2032]	.010”	.008”[.2032]	.008”[.2032]	.005”
Width (3oz)	.014”	.010”	.012”	.010”	.010”	.008”[.2032]
MFG AR (SS/DS)		.006”[.1524]		.004”		.002”
MFG AR (ML)		.009”		.006”[.1524]		.003”
Pad Dia. Via (SS/DS)	Hole + .024”	Hole + .022”	Hole + .020”	Hole + .018”	Hole + .016”	Hole + .014”
Pad Dia. Via (ML)	Hole + .030”	Hole + .028”	Hole + .024”	Hole + .022”	Hole + .018”	Hole + .016”
Pad Dia. Soldered –PLTH (Ideal)	2 X Hole		2 X Hole		2 X Hole	
Pad Dia. Soldered –PLTH (Mid)	1.75 X Hole		1.75 X Hole		1.75 X Hole	
Pad Dia. Soldered –PLTH (Min)	1.5 X Hole		1.5 X Hole		1.5 X Hole	

Note: These values are based on the technology table in the DFM section. Adjust them to accommodate individual findings. The “Use” column is suggested values to use. It is recommended NOT to use the minimums unless necessary.

Table 12: Technology quick-change table

Technology Change	Width reduction	Clearance reduction	Pad reduction (MFG AR)	Aspect ratio reduction	Via pad reduction	Layer increase
Conventional to Advanced	.001" (.006" to .005")	.001" (.006" to .005")	-.003"	8:1 to 10:1	-.003" or new via	10/12 to 20 layers
Advanced to leading edge	.002" (.005" to .003")	.002" (.005" to .003")	-.002"	10:1 to 12:1	-.002" or new via	20 layers to 30 layers

Table 13: Soldered pad diameter

Finished Hole Size	X 1.5	= "Minimum Finished soldered pad diameter"	(no less then Finished hole + AR +MFG AR)
Finished Hole Size	X 1.75	= "Median Finished soldered pad diameter"	(no less then Finished hole + AR +MFG AR)
Finished Hole Size	X 2	= "Maximum Finished soldered pad diameter"	(no less then Finished hole + AR +MFG AR)

Table 14: Annular ring w/o MFG AR

	Plating	+	Spec. AR	=	Min. AR
External Pad	.0025"	+	.002"	=	.0045 round to .005"
Internal Pad	.0025"	+	.001"	=	.0035 round to .004"

Table 15: MFG AR Quick Table per technology

	Min. AR	**+**	**MFG. AR (Conventional/Advanced/ Leading Edge)**			**X**	**2**	**=**	**Pad Area (Conventional/Advanced/ Leading Edge)**		
			Conv.	*Adv*	*LE*				*Conv.*	*Adv*	*LE*
External (DS)	.005	+	.006”	.003”	.002”	X	2	=	.017”	.011”	.009”
External (ML)	.005	+	.009”	.006”	.003”	X	2	=	.023”	.017”	.011”
Internal (DS)	.004	+	.006”	.003”	.002”	X	2	=	.016”	.010”	.008”
Internal (ML)	.004	+	.009”	.006”	.003”	X	2	=	.022”	.016”	.010”

Table 16: Quick pad diameter table (conventional/advanced/leading edge)

	Pad Area	+	Hole Dia.	=	Finished Pad
External (DS)	.017/.011/.009”	+	.031”	=	.048/.042/.040”
External (ML)	.023/.017/.011”	+	.031”	=	.054/.048/.042”
Internal (DS)	.016/.010/.008”	+	.031”	=	.047/.041/.039”
Internal (ML)	.022/.016/.010”	+	.031”	=	.053/.047/.041”

Table 17: Calculating finished pad diameter

AR	+	Mfg AR	* 2	+ Hole	= Pad Diameter
.0025”	+	.009”	* 2		
.0025”	+	.009”	* 2		

Table 18: Material thickness to router bit

Normal FR4	
Board Thickness	*Slot (Router bit)*
.030”	.032” (.031”)
.040” - .080”	.064” (.062”)
.081” >	.083” (.080”)

射 频 和 天 线 设 计 培 训 课 程 推 荐

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立，致力并专注于微波、射频、天线设计研发人才的培养；我们于 2006 年整合合并微波 EDA 网(www.mweda.com)，现已发展成为国内最大的微波射频和天线设计人才培养基地，成功推出多套微波射频以及天线设计经典培训课程和 ADS、HFSS 等专业软件使用培训课程，广受客户好评；并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书，帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司，以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

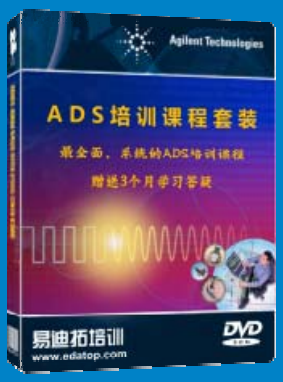
易迪拓培训课程列表：http://www.edatop.com/peixun/rfe/129.html

射频工程师养成培训课程套装

该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材；旨在引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和研发设计能力。通过套装的学习，能够让学员完全达到和胜任一个合格的射频工程师的要求…

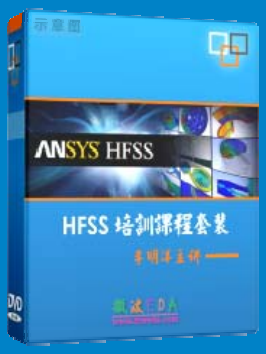
课程网址：http://www.edatop.com/peixun/rfe/110.html

ADS 学习培训课程套装



该套装是迄今国内最全面、最权威的 ADS 培训教程，共包含 10 门 ADS 学习培训课程。课程是由具有多年 ADS 使用经验的微波射频与通信系统设计领域资深专家讲解，并多结合设计实例，由浅入深、详细而又全面地讲解了 ADS 在微波射频电路设计、通信系统设计和电磁仿真设计方面的内容。能让您在最短的时间内学会使用 ADS，迅速提升个人技术能力，把 ADS 真正应用到实际研发工作中去，成为 ADS 设计专家...

课程网址： http://www.edatop.com/peixun/ads/13.html



HFSS 学习培训课程套装

该套课程套装包含了本站全部 HFSS 培训课程，是迄今国内最全面、最专业的 HFSS 培训教程套装，可以帮助您从零开始，全面深入学习 HFSS 的各项功能和在多个方面的工程应用。购买套装，更可超值赠送 3 个月免费学习答疑，随时解答您学习过程中遇到的棘手问题，让您的 HFSS 学习更加轻松顺畅…

课程网址：http://www.edatop.com/peixun/hfss/11.html

CST 学习培训课程套装

该培训套装由易迪拓培训联合微波 EDA 网共同推出，是最全面、系统、专业的 CST 微波工作室培训课程套装，所有课程都由经验丰富的专家授课，视频教学，可以帮助您从零开始，全面系统地学习 CST 微波工作的各项功能及其在微波射频、天线设计等领域的设计应用。且购买该套装，还可超值赠送 3 个月免费学习答疑…

课程网址：http://www.edatop.com/peixun/cst/24.html

HFSS 天线设计培训课程套装

套装包含 6 门视频课程和 1 本图书，课程从基础讲起，内容由浅入深，理论介绍和实际操作讲解相结合，全面系统的讲解了 HFSS 天线设计的全过程。是国内最全面、最专业的 HFSS 天线设计课程，可以帮助您快速学习掌握如何使用 HFSS 设计天线，让天线设计不再难…

课程网址：http://www.edatop.com/peixun/hfss/122.html

13.56MHz NFC/RFID 线圈天线设计培训课程套装

套装包含 4 门视频培训课程，培训将 13.56MHz 线圈天线设计原理和仿真设计实践相结合，全面系统地讲解了 13.56MHz 线圈天线的工作原理、设计方法、设计考量以及使用 HFSS 和 CST 仿真分析线圈天线的具体操作，同时还介绍了 13.56MHz 线圈天线匹配电路的设计和调试。通过该套课程的学习，可以帮助您快速学习掌握 13.56MHz 线圈天线及其匹配电路的原理、设计和调试…

详情浏览：http://www.edatop.com/peixun/antenna/116.html

我们的课程优势：

※ 成立于 2004 年，10 多年丰富的行业经验，

※ 一直致力并专注于微波射频和天线设计工程师的培养，更了解该行业对人才的要求

※ 经验丰富的一线资深工程师讲授，结合实际工程案例，直观、实用、易学

联系我们：

※ 易迪拓培训官网：http://www.edatop.com

※ 微波 EDA 网：http://www.mweda.com

※ 官方淘宝店：http://shop36920890.taobao.com