Design for Manufacurability

1. Design for Fabrication

1. Padstacks

type	pitch	hole over Max pin	pad over hole round up to the next .005	Anti/therm over hole	soldermask over pad	solderpaste over pad	notes
thru-hole	all	.015	.020	.030	.010	n/a	
smd	≥.65mm	n/a	ipc-sm-782a	n/a	.010	same	
smd	between	n/a	ipc-sm-782a	n/a	.006	same	
smd	≤.40mm	n/a	ipc-sm-782a	n/a	.002	same	gang mask IC pads

2. Trace Widths
Minimum trace width – prefered .012 – acceptable .006

3. Spacings
Trace – Trace Minimum – preferred .012 – acceptable .006
Pad – Pad Minimum – preferred .015 – acceptable .013
Pad – Trace Minimum – preferred .010 – acceptable .008
(Traces include all conductors covered in Solder Mask – Pads include vias and all exposed conductors)
no copper within .025 from edge of PCB

4. Board Thicknesses
Standard Board thicknesses are .031, .062,.093 & .125

5. Board sizes
Try to base Board sizes on a raw panel size of 24x18

2. Design for Assembly

1. Assembly tooling holes
.125 dia. Holes added in 3 corners minimum.

2. fiducials
3 fiducials added to the board for SMD pick and place machine
local fine pitch SMD IC fiducials added to the board as needed

3. Component Selection
SMD components should be used whenever possible
Auto-insertable components should be used whenever possible

4. Component Placement
All components on 1 side preferable
All like components oriented in the same direction preferable
SMD components .200 from edge of PCB
T/H components .050 from edge of PCB

3. Design for Test

1. Testing tooling holes
.125 dia. Holes added in 2 opposite corners.
(may be combined with assembly tooling holes)

2. Test points
All nets to have bottom side test points for flying lead
Or bed of nails testing