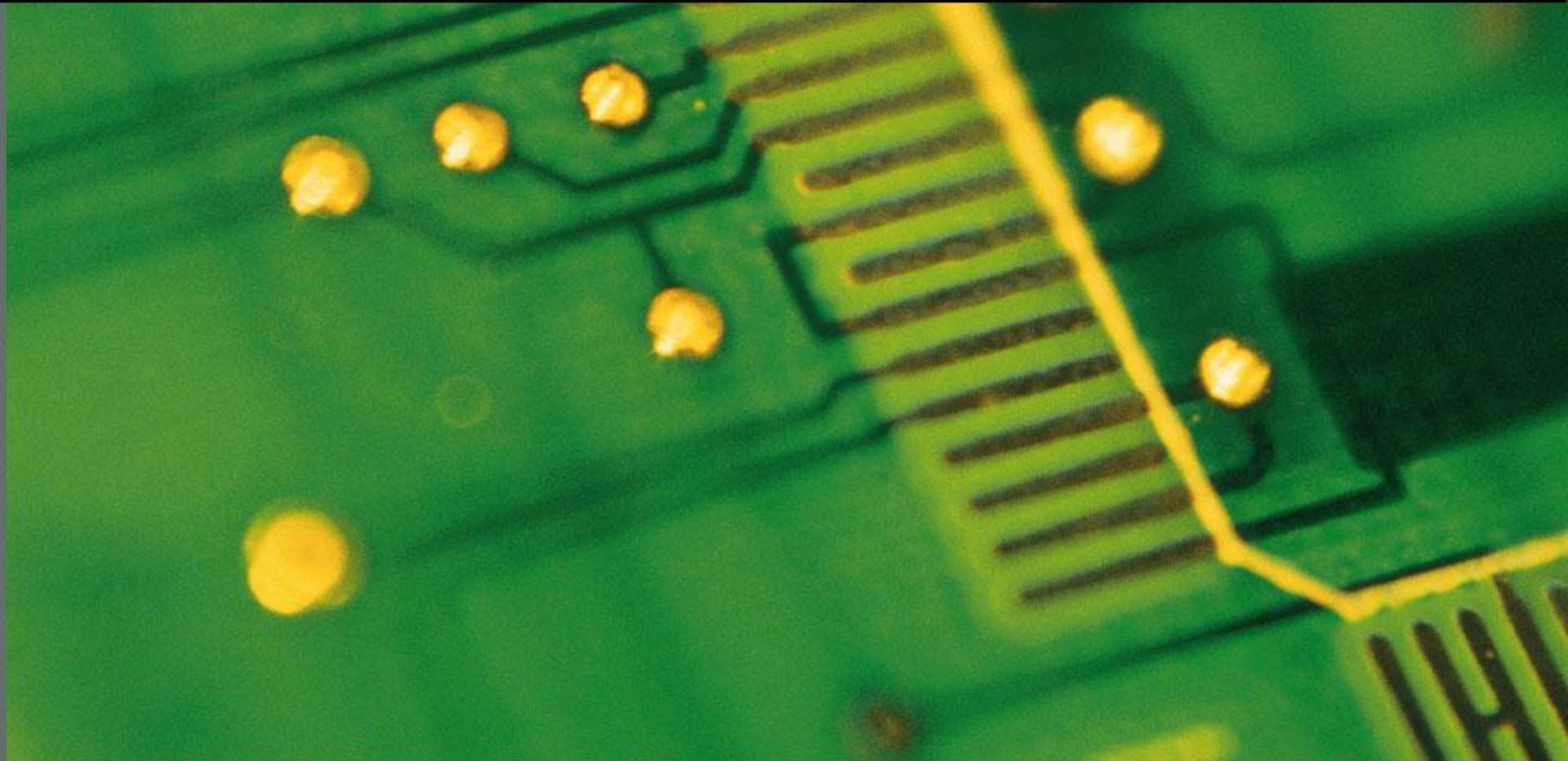


One Way Pcb Ltd

Printed Circuit Board Solutions Provider



Front End DRC

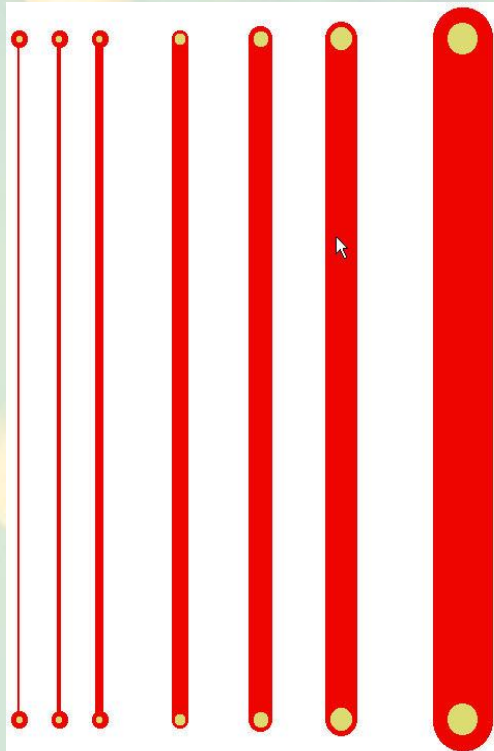


- **Presentation Title:**
Front End DRC (Design Rule Checks).
- **Created / Modified:**
April 2018.
- **Compiled By:**
One Way Pcb Ltd.
- **Description:**
An introduction to Design Rule Checks.
The following is a guide only.

Minimum Track Width



- This determines the minimum track width used on the board.



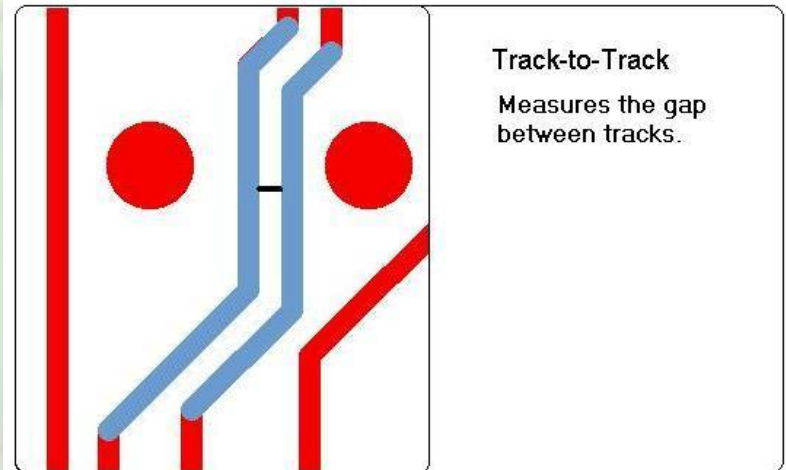
Minimum Conductor Width						
	Copper	Plate to	Good	Batch	R & D	Mil /
		25micron	Practice	Minimum	Minimum	IPC Class III
1	12 micron (0.33oz)	No	5	3.5	3	As Standard
2	17 micron (1/2oz)	No	6	4	3.5	As Standard
3	35 micron (1oz)	No	7	5	4.5	As Standard
4	70 micron (2oz)	No	10	8	7	As Standard
5	105 micron (3oz)	No	13	11	10	As Standard
6	140 micron (4oz)	No	16	14	13	As Standard

Feature to feature spacing



- This determines the minimum gap between any two features e.g. Track to track, pad to track, pad to pad, plane to pad etc.
- This shows the track to track spacing, or 'gaps'.

Feature to Feature Spacing						
Copper	Plate to	Good	Batch	R & D	Mil /	IPC Class III
	25micron	Practice	Minimum	Minimum		
1	12 micron (0.33oz)	No	5.5	4.5	4	As Standard
2	17 micron (1/2oz)	No	6	5	4	As Standard
3	35 micron (1oz)	No	7	6	5	As Standard
4	70 micron (2oz)	No	9	8	7.5	As Standard
5	105 micron (3oz)	No	11	10	9.5	As Standard
6	140 micron (4oz)	No	13	12	11.5	As Standard

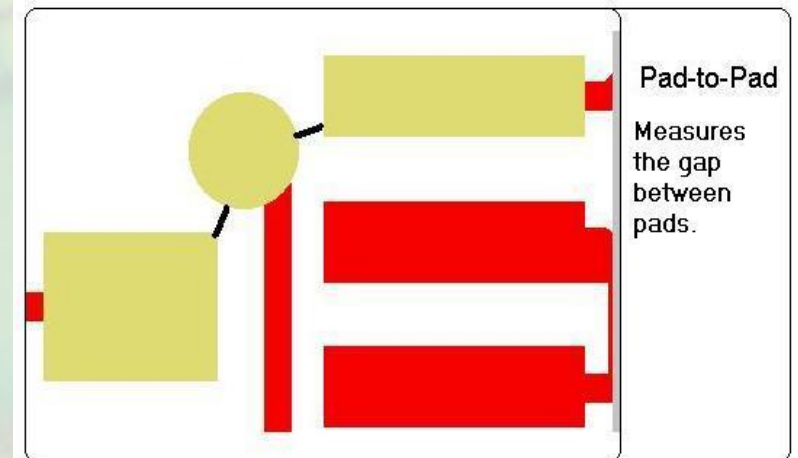


Feature to feature spacing



- Pad to pad.
- DRC finds the minimum space (gap) between any pads.

Feature to Feature Spacing						
Copper	Plate to	Good	Batch	R & D	Mil /	
	25micron	Practice	Minimum	Minimum	IPC Class III	
1	12 micron (0.33oz)	No	5.5	4.5	4	As Standard
2	17 micron (1/2oz)	No	6	5	4	As Standard
3	35 micron (1oz)	No	7	6	5	As Standard
4	70 micron (2oz)	No	9	8	7.5	As Standard
5	105 micron (3oz)	No	11	10	9.5	As Standard
6	140 micron (4oz)	No	13	12	11.5	As Standard

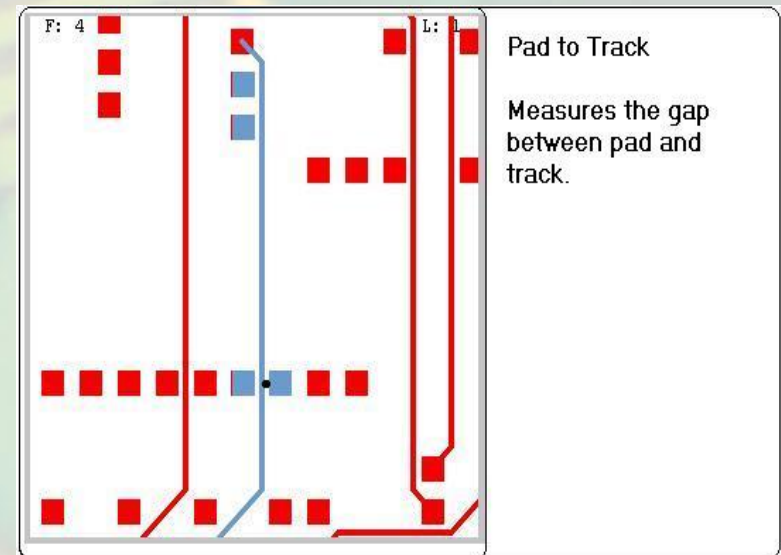


Feature to feature spacing



- Pad to track.
- DRC determines the spaces between any pad and track. (Note Poor Auto routing)

Feature to Feature Spacing						
Copper	Plate to	Good	Batch	R & D	Mil /	
	25micron	Practice	Minimum	Minimum	IPC Class III	
1	12 micron (0.33oz)	No	5.5	4.5	4	As Standard
2	17 micron (1/2oz)	No	6	5	4	As Standard
3	35 micron (1oz)	No	7	6	5	As Standard
4	70 micron (2oz)	No	9	8	7.5	As Standard
5	105 micron (3oz)	No	11	10	9.5	As Standard
6	140 micron (4oz)	No	13	12	11.5	As Standard



Minimum Annular Ring



- There are many standards applied to the minimum annular ring analysis, e.g. MIL Spec, buried/blind vias, copper weight, all of which determine the minimum requirement.

Minimum Annular Ring (Not Landless vias) Single Bond					
Copper	Plate to	Good	Batch	R & D	Mil /
	25micron	Practice	Minimum	Minimum	IPC Class III
1 12 micron (0.33oz)	No	8	6	5	12
2 17 micron (1/2oz)	No	8	6	5	12
3 35 micron (1oz)	No	9	7	6	12
4 70 micron (2oz)	No	12	10	8	12
5 105 micron (3oz)	No	14	12	10	14
6 140 micron (4oz)	No	16	14	12	16



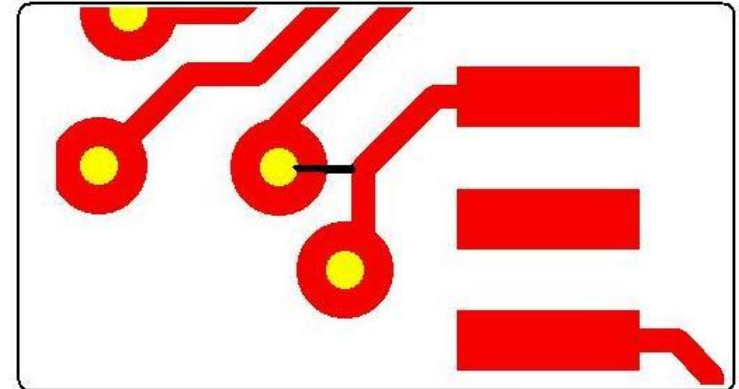
Annular Ring

Measures the annular 'ring' of copper around a drilled hole.

Drilled Hole to Feature



- This is an important requirement, to prevent errors due to 'drill wander' and misalignment.



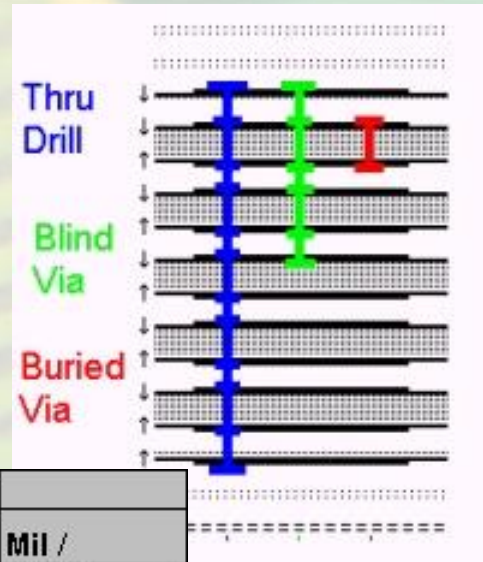
Drilled Hole-to-Feature(pad, track, plane)
Measures the distance from the edge of a drilled hole (hole shown as yellow) to the nearest feature. i.e. pad, track, plane.

Minimum Drilled Hole to Feature					
Feature	Plate to	Good	Batch	R & D	Mil /
	25micron	Practice	Minimum	Minimum	IPC Class III
1 Inner Layer Main Drill	No	12	10	9	12
2 Inner Via Layers Via + Main Drill	Yes	12	10	9	12
3 Outer / Plated Layers Through Package Drill	Yes	10	8.5	7.5	12
4 Multi-Bond Layers Via Drill (first bond)	No	12	10	9	12
5 Multi-Bond Layers Main / Via Drill (and if more than 1 bond)	No	15	13	12	14

Annular Ring & Drilled Hole to Feature on Multibond Packages



- Since material moves with each bonding operation, Annular ring and drilled hole to feature distance must be greater on multi-bond packages



Multi-bond rules apply

Single bond rules apply

Minimum Drilled Hole to Feature

Feature	Plate to 25micron	Good Practice	Batch Minimum	R & D Minimum	Mil / IPC Class III
Inner Layer Main Drill	No	12	10	9	12
Inner Via Layers Via + Main Drill	Yes	12	10	9	12
Outer / Plated Layers Through Package Drill	Yes	10	8.5	7.5	12
Multi-Bond Layers Via Drill (first bond)	No	12	10	9	12
Multi-Bond Layers Main / Via Drill (and if more than 1 bond)	No	15	13	12	14

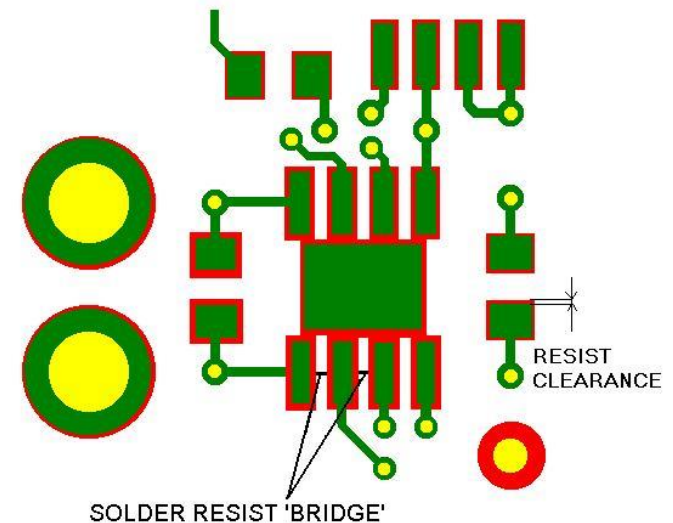
Solder Resist Clearance



- The solder resist 'bridge' serves to prevent solder shorts between surface mount pads.
- It is also important to provide enough clearance to prevent soldermask encroachment onto features.

Other guidelines					
		Minimum Requirements			Mil /
		Good	Batch	R & D	IPC Class III
Soldermask Bridge		4	3 ***	2.5 ***	As standard
Soldermask Clearance		4	3 ***	2.5 ***	As standard
S/Mask Clearance BGA	Required	3.5	3 ***	2.5 ***	As standard
Legend Line Width		8	7	6	As standard
Copper to Board Edge	All Layers	25	15	10	25
Copper to Board Edge Score / Chamfer	Check Chamfer Angle and Write check requirement in box and check.				
Plane to Feature Gap	Plated Layers	10	9	8	As standard

SOLDER RESIST CLEARANCE



Solder Resist Bridge



- The solder resist 'bridge' prevents solder shorts between features. But if it is too narrow, it may lift and become detached. It is preferable to remove the bridge from the data in these circumstance.

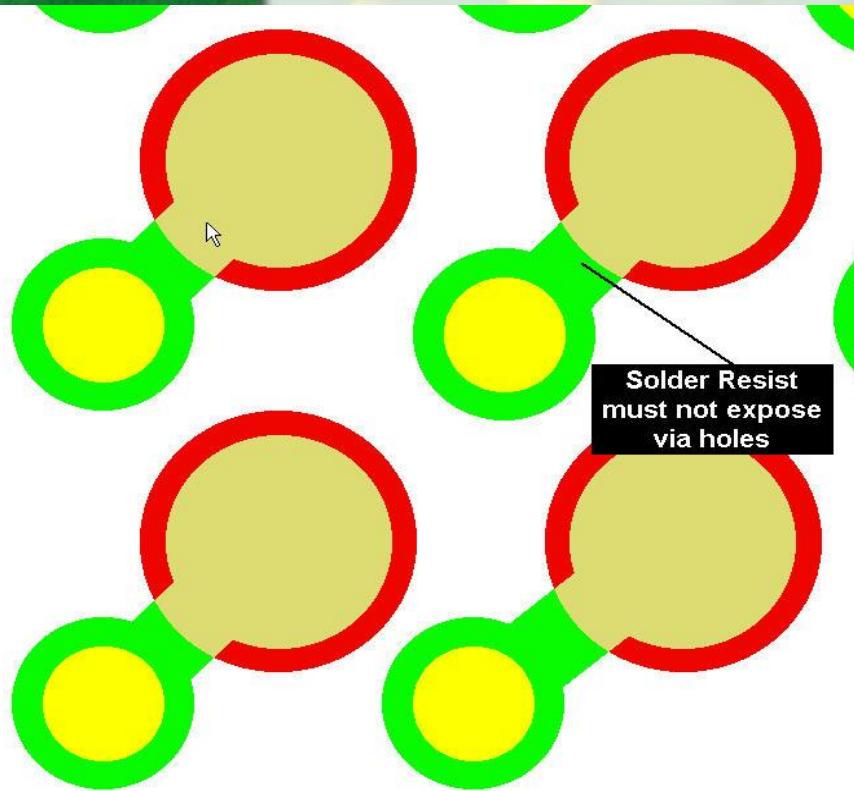
Other guidelines					
		Minimum Requirements			Mil /
		Good	Batch	R & D	IPC Class III
Soldermask Bridge		4	3 ***	2.5 ***	As standard
Soldermask Clearance		4	3 ***	2.5 ***	As standard
S/Mask Clearance BGA	Required	3.5	3 ***	2.5 ***	As standard
Legend Line Width		8	7	6	As standard
Copper to Board Edge	All Layers	25	15	10	25
Copper to Board Edge Score / Chamfer	Check Chamfer Angle and Write check requirement in box and check.				
Plane to Feature Gap	Plated Layers	10	9	8	As standard



Solder Resist Clearance BGA



- On BGA's (Ball Grid Array) the solder resist clearance must NOT expose the via holes.

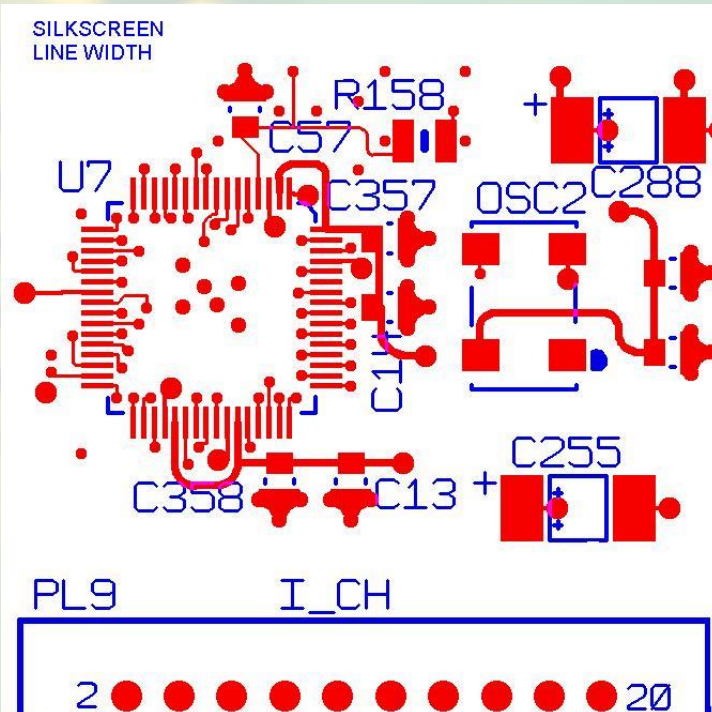


Other guidelines					
		Minimum Requirements			Mil /
		Good	Batch	R & D	IPC Class III
Soldermask Bridge		4	3 ***	2.5 ***	As standard
Soldermask Clearance		4	3 ***	2.5 ***	As standard
S/Mask Clearance BGA	Required	3.5	3 ***	2.5 ***	As standard
Legend Line Width		8	7	6	As standard
Copper to Board Edge	All Layers	25	15	10	25
Copper to Board Edge Score / Chamfer	Check Chamfer Angle and Write check requirement in box and check.				
Plane to Feature Gap	Plated Layers	10	9	8	As standard

Legend Line Width



- Any legend that is on a pad/surface mount pad is removed.
- The legend line width should not be too small, as it will not print clearly.



Other guidelines					
		Minimum Requirements			Mil / IPC Class III
		Good	Batch	R & D	
Soldermask Bridge		4	3 ***	2.5 ***	As standard
Soldermask Clearance		4	3 ***	2.5 ***	As standard
S/Mask Clearance BGA	Required	3.5	3 ***	2.5 ***	As standard
Legend Line Width		8	7	6	As standard
Copper to Board Edge	All Layers	25	15	10	25
Copper to Board Edge Score / Chamfer	Check Chamfer Angle and Write check requirement in box and check.				
Plane to Feature Gap	Plated Layers	10	9	8	As standard

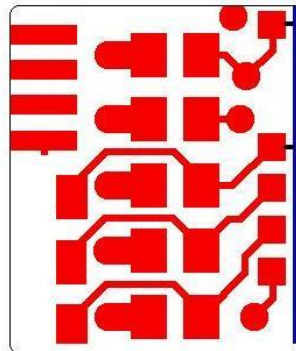
Copper to Board Edge



- All copper features such as pads, tracks or ground planes should be clear of the board edge.

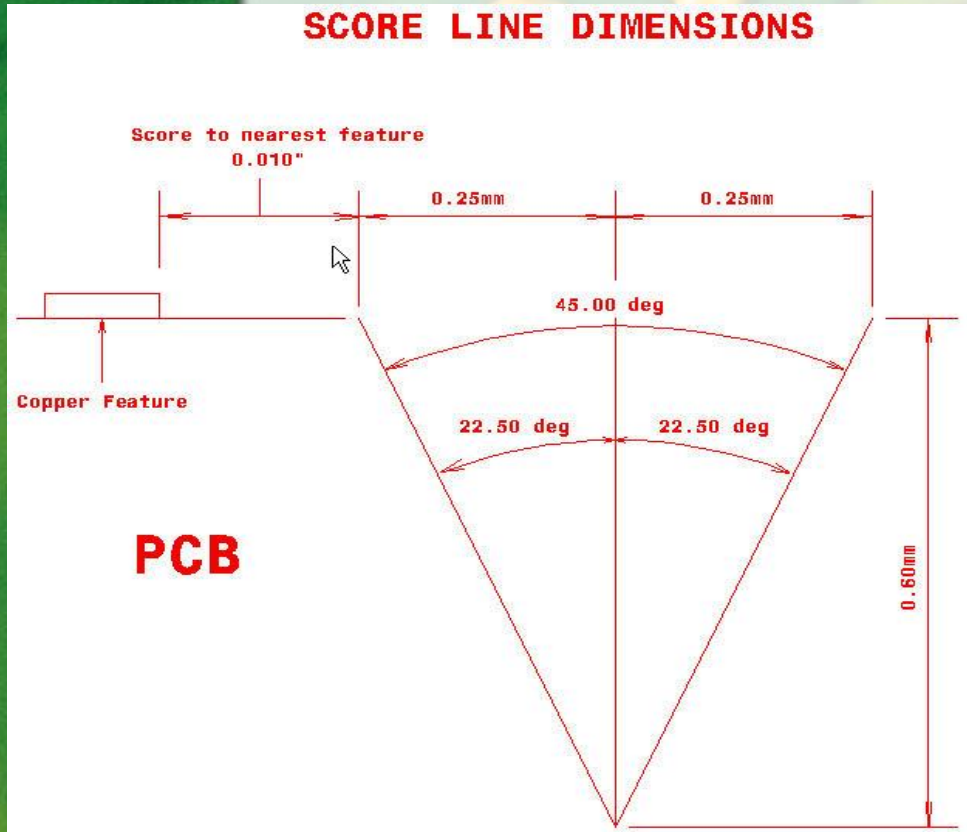
Copper to Board Edge

The distance of the nearest copper feature (pad, track, ground plane) to the board profile edge.



Other guidelines					
		Minimum Requirements			Mil /
		Good	Batch	R & D	IPC Class III
Soldermask Bridge		4	3 ***	2.5 ***	As standard
Soldermask Clearance		4	3 ***	2.5 ***	As standard
S/Mask Clearance BGA	Required	3.5	3 ***	2.5 ***	As standard
Legend Line Width		8	7	6	As standard
Copper to Board Edge	All Layers	25	15	10	25
Copper to Board Edge Score / Chamfer	Check Chamfer Angle and Write check requirement in box and check.				
Plane to Feature Gap	Plated Layers	10	9	8	As standard

Copper to Board Edge (Score)



- These dimensions should always be checked against the customers requirements.

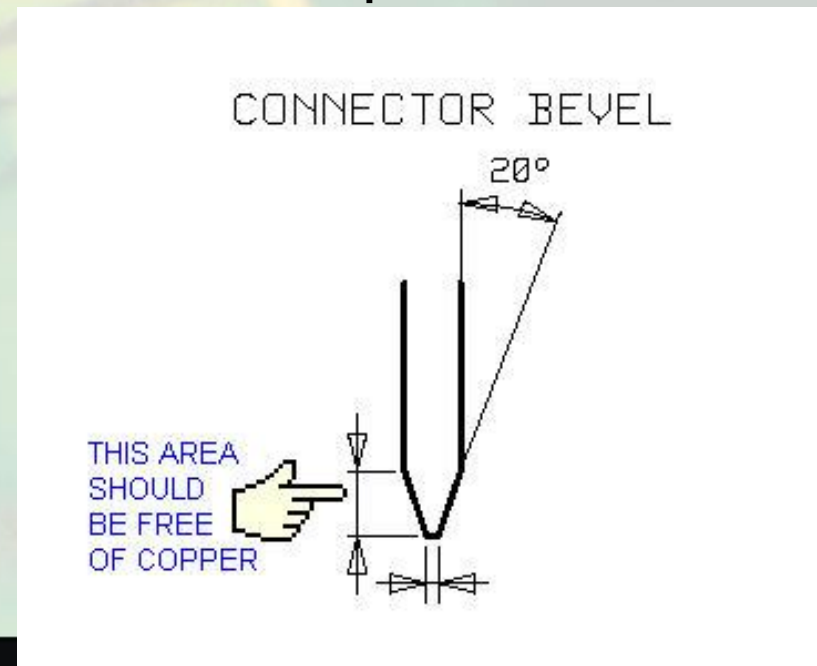
Other guidelines					
		Minimum Requirements			Mil /
		Good	Batch	R & D	IPC Class III
Soldermask Bridge		4	3 ***	2.5 ***	As standard
Soldermask Clearance		4	3 ***	2.5 ***	As standard
S/Mask Clearance BGA	Required	3.5	3 ***	2.5 ***	As standard
Legend Line Width		8	7	6	As standard
Copper to Board Edge	All Layers	25	15	10	25
Copper to Board Edge Score / Chamfer	Check Chamfer Angle and Write check requirement in box and check.				
Plane to Feature Gap	Plated Layers	10	9	8	As standard

Copper to Board Edge (chamfer)



- During routing, scoring or chamfering the copper if too close to the edge may be damaged, ripped off or cause other problems.
- Check dimensions with customer requirements.

Other guidelines					
		Minimum Requirements			Mil /
		Good	Batch	R & D	IPC Class III
Soldermask Bridge		4	3 ***	2.5 ***	As standard
Soldermask Clearance		4	3 ***	2.5 ***	As standard
S/Mask Clearance BGA	Required	3.5	3 ***	2.5 ***	As standard
Legend Line Width		8	7	6	As standard
Copper to Board Edge	All Layers	25	15	10	25
Copper to Board Edge Score / Chamfer	Check Chamfer Angle and Write check requirement in box and check.				
Plane to Feature Gap	Plated Layers	10	9	8	As standard

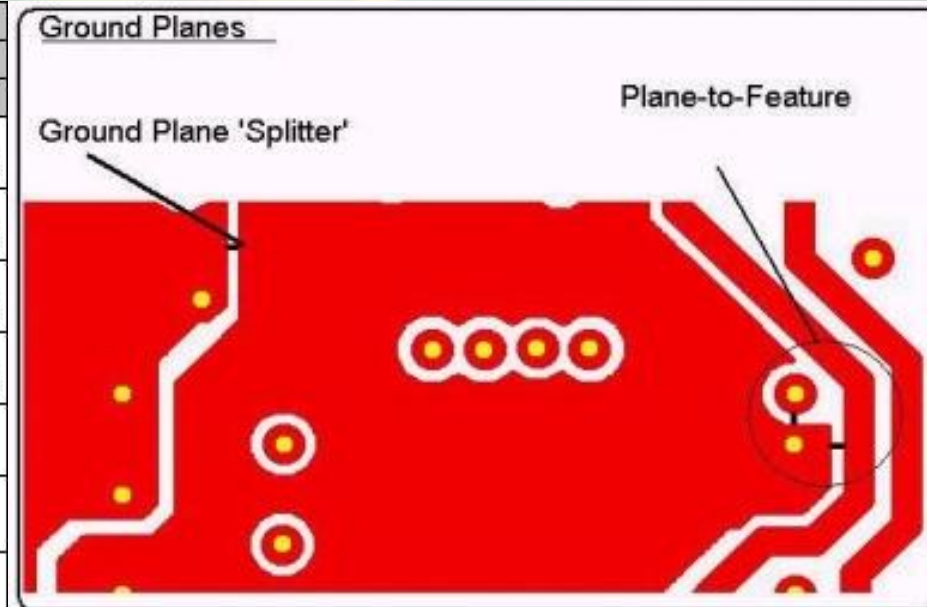


Ground Plane Features



- **Ensure adequate clearance between plane and feature. Particularly important on plated copper planes.**

	Other guidelines				
		Minimum Requirements			Mil /
		Good	Batch	R & D	IPC Class III
Soldermask Bridge		4	3 ***	2.5 ***	As standard
Soldermask Clearance		4	3 ***	2.5 ***	As standard
S/Mask Clearance BGA	Required	3.5	3 ***	2.5 ***	As standard
Legend Line Width		8	7	6	As standard
Copper to Board Edge	All Layers	25	15	10	25
Copper to Board Edge Score / Chamfer	Check Chamfer Angle and Write check requirement in box and check.				
Plane to Feature Gap	Plated Layers	10	9	8	As standard

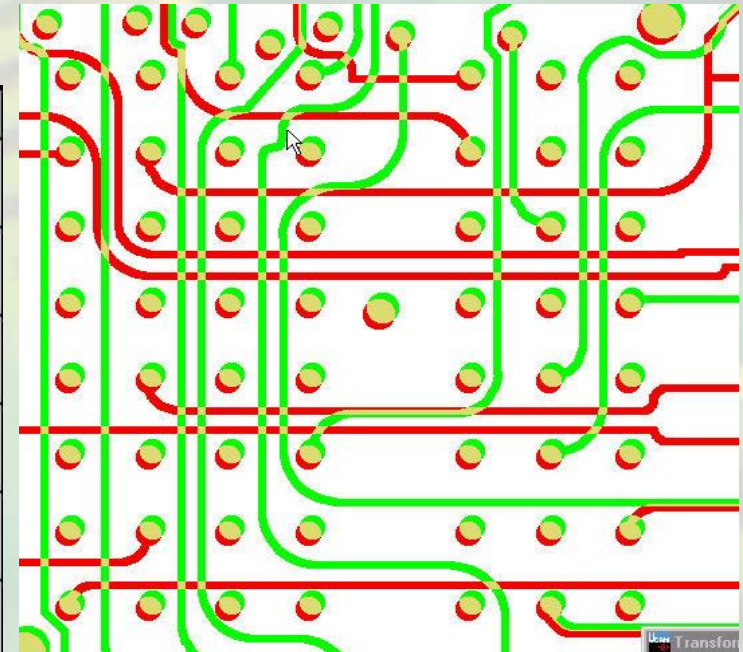


Layer to Layer Registration



- All the layers must be properly registered.
- Any mis-registration could lead to further problems during the DRC.

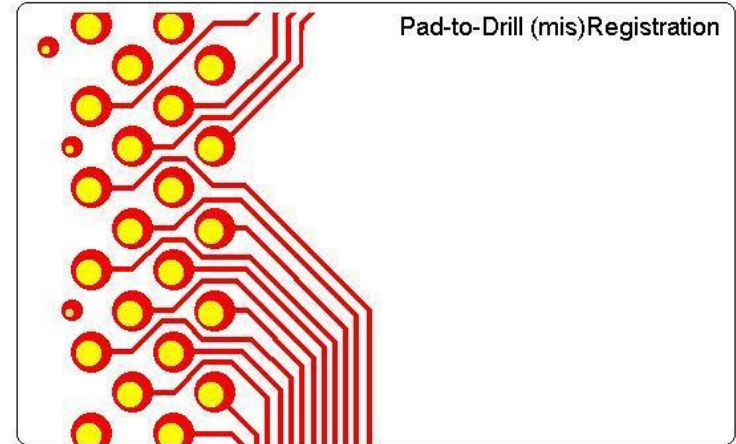
	Other Checks	Tick Below
Layer to Layer Registration	Tick To confirm checks completed and ok	<input type="checkbox"/>
Pad to Drill Registration	Tick To confirm checks completed and ok	<input type="checkbox"/>
Missing Holes / Pads	Tick To confirm checks completed and ok	<input type="checkbox"/>
Blind / Buried Vias	Tick To confirm checks completed and ok	<input type="checkbox"/>
Pad/Drill Connections	Tick To confirm checks completed and ok	<input type="checkbox"/>
Net List Compare Completed	Tick To confirm checks completed and ok	<input type="checkbox"/>
Net List Compare Completed after any Mods	Tick To confirm checks completed and ok	<input type="checkbox"/>



Pad to Drill Registration



- Registration must be in the centre of the pad.
- At this stage, any misregistration could lead to further problems during the DRC



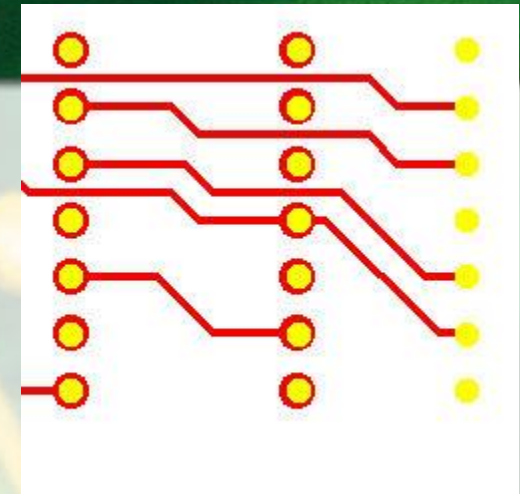
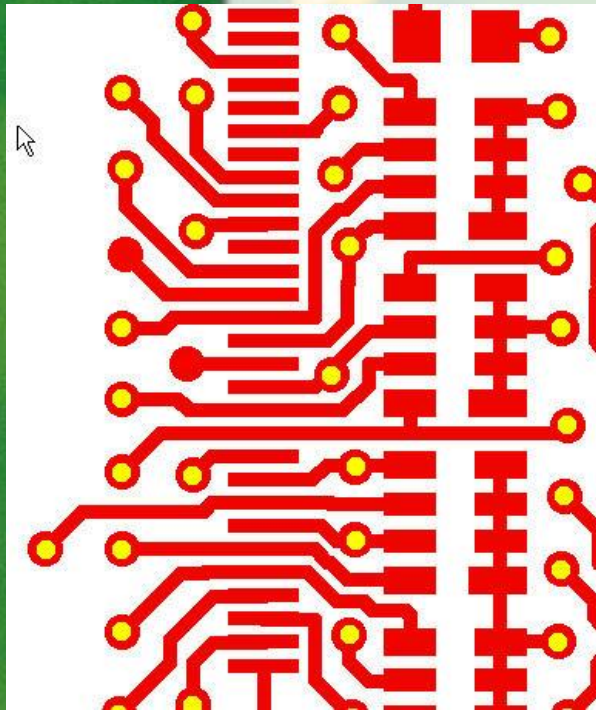
AN EXAMPLE OF MISREGISTRATION OF THE DRILLS TO PADS.

Other Checks		Tick Below
Layer to Layer Registration	Tick To confirm checks completed and ok	
Pad to Drill Registration	Tick To confirm checks completed and ok	
Missing Holes / Pads	Tick To confirm checks completed and ok	
Blind / Buried Vias	Tick To confirm checks completed and ok	
Pad/Drill Connections	Tick To confirm checks completed and ok	
Net List Compare Completed	Tick To confirm checks completed and ok	
Net List Compare Completed after any Mods	Tick To confirm checks completed and ok	

Missing Holes/Pads



- Spot the obvious!

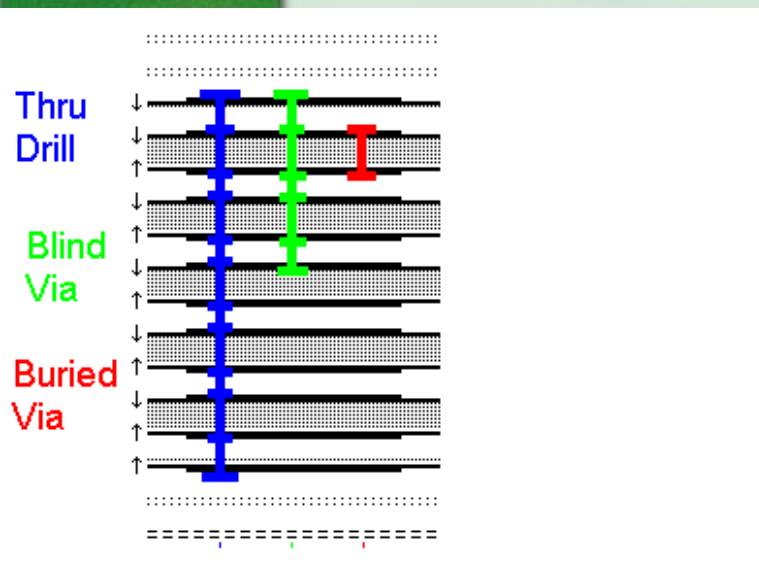


Other Checks		Tick Below
Layer to Layer Registration	Tick To confirm checks completed and ok	
Pad to Drill Registration	Tick To confirm checks completed and ok	
Missing Holes / Pads	Tick To confirm checks completed and ok	
Blind / Buried Vias		
Pad/Drill Connections	Tick To confirm checks completed and ok	
Net List Compare Completed	Tick To confirm checks completed and ok	
Net List Compare Completed after any Mods	Tick To confirm checks completed and ok	

Blind/Buried Vias Pad/Drill Connections



- Check there is a pad (connection) for the termination of each drill (via).

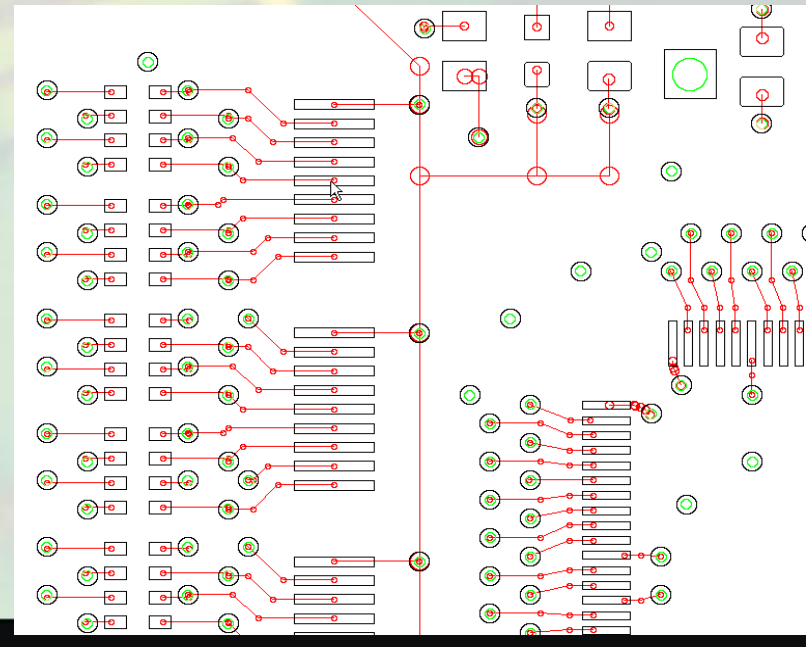
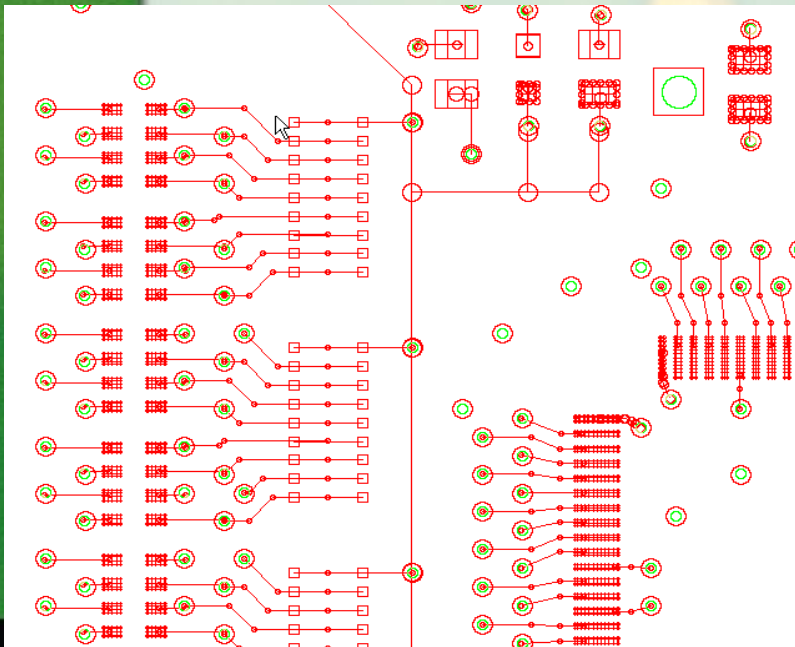


Other Checks		Tick Below
Layer to Layer Registration	Tick To confirm checks completed and ok	
Pad to Drill Registration	Tick To confirm checks completed and ok	
Missing Holes / Pads	Tick To confirm checks completed and ok	
Blind / Buried Vias		
Pad/Drill Connections	Tick To confirm checks completed and ok	
Net List Compare Completed	Tick To confirm checks completed and ok	
Net List Compare Completed after any Mods	Tick To confirm checks completed and ok	

Drawn Data



- Drawn pads must be replaced with 'flashed' pads.
- Modelling replaces the 'draws' with a single pad of the exact size and shape.
- On this job 13,770 'draws' were replaced by 1,312 pads!



Drawn Data



- Drawn planes created as multiple fine line draws should be replaced with 'contourised data'.
- This significantly reduces file size and speeds up data processing.

