

- Notes:
- Board shall be fabricated - performance class II as per IPC-6011 and IPC6012
 - PCB manufacturer logo, P/N, revision and/or date code of manufacturing shall be printed in top solder mask (not over pcb traces, allowed over copper plane).
The date code shall be in the format: "WWYY" where WW=week and YY= year, max height 0.15 inches
 - Silkscreen printed on both sides
 - Material: high temperature FR4 class epoxy glass rated UL94V-0. UL symbol and rating shall be marked farside
35um copper for external layers and 18um for all internal layers
Must be RoHS compliant and survive a lead-free assembly max reflow of 260 deg C (5 passes)
Td rating: >340 deg C
Tg = 150 deg C (min)
 - Solder mask: SMOBC per IPC-SM-840C, class T must be Rohs compliant, 0.001" max measured over bare copper plating, must clear all lands as indicated on gerber solder mask layers, color= GREEN
 - Finish: electro-less nickel immersion gold (ENIG), 0.05-0.125um Au over 3-6um Ni - over bare copper only
 - Solderability test: Category 2 of J-STD-003
 - Finished boards shall not have nicks, scratches, voids, exposed copper, poor plating or misdrilled holes
 - All holes sizes are after plating
 - PCB manufacturer may add copper thieving as needed to improve manufacturability, thieving to be 0.030" round pads at 0.050" spacing.
Thieving will have a minimum of 0.100" clearance from existing copper and should not be placed under surface mounted devices
 - PCB manufacturer may use tear drops to improve annular rings as long as DRC rules are followed
 - All via connections to power and ground planes are solid
 - All unconnected pads on inner signal layers are removed
 - All finished boards are to be 100% electrically tested
 - Unless otherwise indicated, all linear tolerances shall be XX.X +/-0.2mm and XX.XX +/- 0.1mm
 - Gerber file GM1 shows board outline (milling line)
 - Table 1 shows Layer stack details

Additional notes:

A1. Finished board thickness = 68.5mils +/- 10%; measured over top/bottom copper and solder mask

A1. Blind vias are from L1 to L2 and L5 to L6

Table 1: Layer Stack Details for IDL_15_38 Rev.A (Imperial Units)

Layer	Name	Material	Thickness	Constant	Board Layer Stack
1	Top Overlay				
2	Top Solder	Solder Resist	1.50mil	3.5	
3	Top Layer - SIG1	Copper	1.38mil		
4	Dielectric 1	FR4	15.00mil	4.65	
5	Layer 2 - GND1	Copper	0.70mil		
6	Dielectric 2	FR4	9.00mil	4.65	
7	Layer 3 - SIG2	Copper	0.70mil		
8	Dielectric 3	FR4	12.00mil	4.65	
9	Layer 4 - PWR	Copper	0.70mil		
10	Dielectric 4	FR4	9.00mil	4.65	
11	Layer 5 - GND2	Copper	0.70mil		
12	Dielectric 5	FR4	15.00mil	4.65	
13	Bottom Layer - SIG3	Copper	1.38mil		
14	Bottom Solder	Solder Resist	1.50mil	3.5	
15	Bottom Overlay				

Table 2: NC Drill Details for IDL_15_38 Rev.A

Symbol	Hil Count	Finished Hole Size	Plated	Hole Type
⊕	2	0.630mm (24.80mil)	PTH	Round
⊗	2	1.450mm (57.09mil)	NPTH	Round
▽	2	1.905mm (75.00mil)	PTH	Round
⊗	4	0.760mm (29.92mil)	NPTH	Round
◇	4	1.000mm (39.37mil)	PTH	Round
⊗	126	2.500mm (98.43mil)	PTH	Round
○	184	0.400mm (15.75mil)	PTH	Round
□	3400	0.300mm (11.81mil)	PTH	Round
3724 Total				



- Notes:
- Board shall be fabricated - performance class II as per IPC-6011 and IPC6012
 - PCB manufacturer logo, P/N, revision and/or date code of manufacturing shall be printed in top solder mask (not over pcb traces, allowed over copper plane). The date code shall be in the format: "WWYY" where WW=week and YY= year, max height 0.15 inches
 - Silkscreen printed on both sides
 - Material: high temperature FR4 class epoxy glass rated UL94V-0. UL symbol and rating shall be marked farside
35um copper for external layers and 18um for all internal layers
Must be RoHS compliant and survive a lead-free assembly max reflow of 260 deg C (5 passes)
Td rating: >340 deg C
Tg = 150 deg C (min)
 - Solder mask: SMOBC per IPC-SM-840C, class T must be Rohs compliant, 0.001" max measured over bare copper plating, must clear all lands as indicated on gerber solder mask layers, color= GREEN
 - Finish: electro-less nickel immersion gold (ENIG), 0.05-0.125um Au over 3-6um Ni - over bare copper only
 - Solderability test: Category 2 of J-STD-003
 - Finished boards shall not have nicks, scratches, voids, exposed copper, poor plating or misdrilled holes
 - All holes sizes are after plating
 - PCB manufacturer may add copper thieving as needed to improve manufacturability, thieving to be 0.030" round pads at 0.050" spacing. Thieving will have a minimum of 0.100" clearance from existing copper and should not be placed under surface mounted devices
 - PCB manufacturer may use tear drops to improve annular rings as long as DRC rules are followed
 - All via connections to power and ground planes are solid
 - All unconnected pads on inner signal layers are removed
 - All finished boards are to be 100% electrically tested
 - Unless otherwise indicated, all linear tolerances shall be XX.X +/-0.2mm and XX.XX +/-0.1mm
 - Gerber file GM1 shows board outline (milling line)
 - Table 1 shows Layer stack details

Additional notes:

A1. Finished board thickness = 68.5mils +/- 10%; measured over top/bottom copper and solder mask

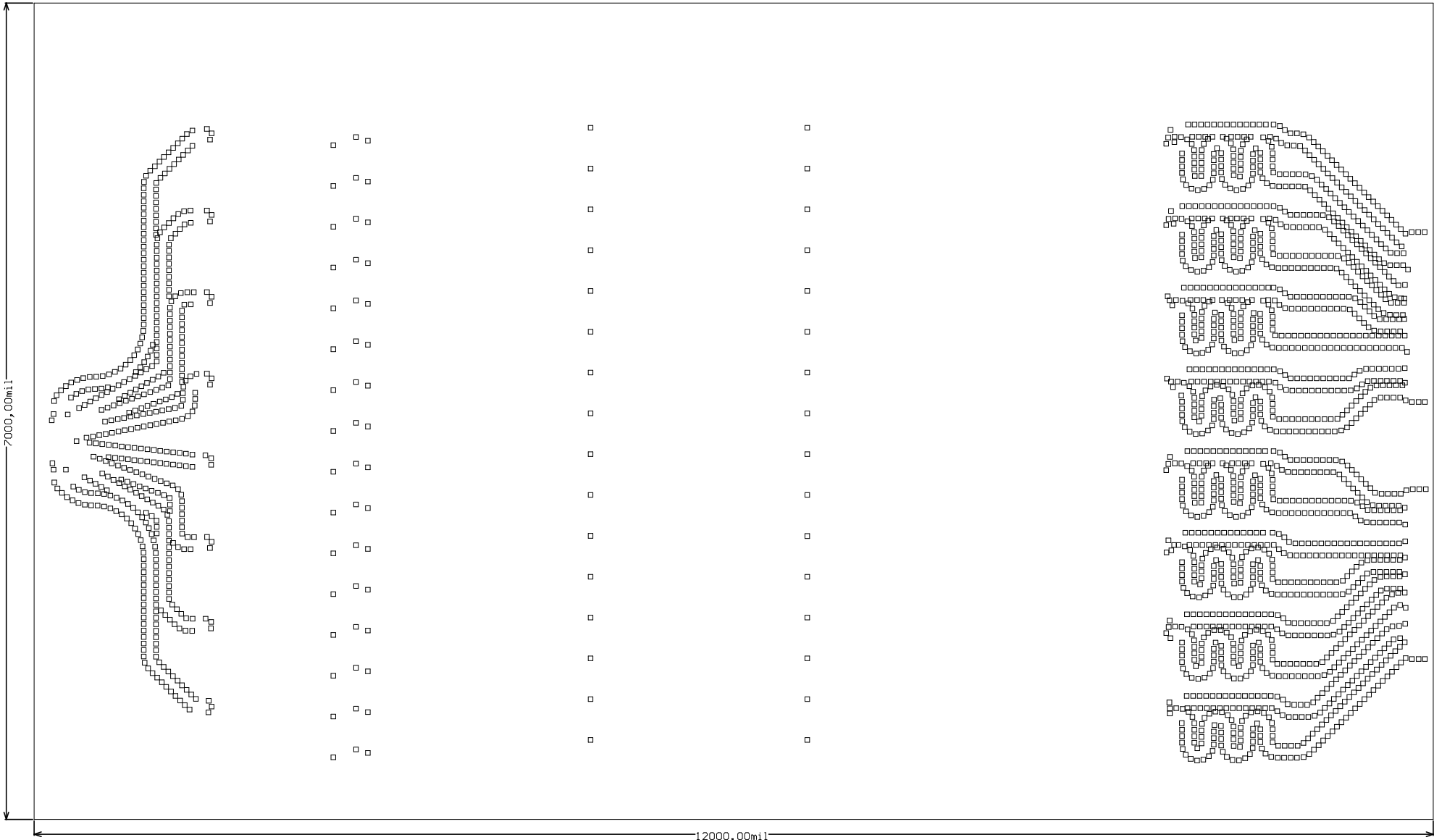
A1. Blind vias are from L1 to L2 and L5 to L6

Table 1: Layer Stack Details for IDL_15_38 Rev.A (Imperial Units)

Layer	Name	Material	Thickness	Constant	Board Layer Stack
1	Top Overlay				
2	Top Solder	Solder Resist	1.50mil	3.5	
3	Top Layer - SIG1	Copper	1.38mil		
4	Dielectric 1	FR4	15.00mil	4.65	
5	Layer 2 - GND1	Copper	0.70mil		
6	Dielectric 2	FR4	9.00mil	4.65	
7	Layer 3 - SIG2	Copper	0.70mil		
8	Dielectric 3	FR4	12.00mil	4.65	
9	Layer 4 - PWR	Copper	0.70mil		
10	Dielectric 4	FR4	9.00mil	4.65	
11	Layer 5 - GND2	Copper	0.70mil		
12	Dielectric 5	FR4	15.00mil	4.65	
13	Bottom Layer - SIG3	Copper	1.38mil		
14	Bottom Solder	Solder Resist	1.50mil	3.5	
15	Bottom Overlay				

Table 2: NC Drill Details for IDL_15_38 Rev.A

Symbol	Hil Count	Finished Hole Size	Plated	Hole Type
□	5335	0.300mm (11.81mil)	PTH	Round
5335 Total				



- Notes:
- Board shall be fabricated - performance class II as per IPC-6011 and IPC6012
 - PCB manufacturer logo, P/N, revision and/or date code of manufacturing shall be printed in top solder mask (not over pcb traces, allowed over copper plane). The date code shall be in the format: "WWYY" where WW=week and YY= year, max height 0.15 inches
 - Silkscreen printed on both sides
 - Material: high temperature FR4 class epoxy glass rated UL94V-0. UL symbol and rating shall be marked farside
35um copper for external layers and 18um for all internal layers
Must be RoHS compliant and survive a lead-free assembly max reflow of 260 deg C (5 passes)
Td rating: >340 deg C
Tg = 150 deg C (min)
 - Solder mask: SMOBC per IPC-SM-840C, class T must be Rohs compliant, 0.001" max measured over bare copper plating, must clear all lands as indicated on gerber solder mask layers, color= GREEN
 - Finish: electro-less nickel immersion gold (ENIG), 0.05-0.125um Au over 3-6um Ni - over bare copper only
 - Solderability test: Category 2 of J-STD-003
 - Finished boards shall not have nicks, scratches, voids, exposed copper, poor plating or misdrilled holes
 - All holes sizes are after plating
 - PCB manufacturer may add copper thieving as needed to improve manufacturability, thieving to be 0.030" round pads at 0.050" spacing. Thieving will have a minimum of 0.100" clearance from existing copper and should not be placed under surface mounted devices
 - PCB manufacturer may use tear drops to improve annular rings as long as DRC rules are followed
 - All via connections to power and ground planes are solid
 - All unconnected pads on inner signal layers are removed
 - All finished boards are to be 100% electrically tested
 - Unless otherwise indicated, all linear toleracnes shall be XX.X +/-0.2mm and XX.XX +/- 0.1mm
 - Gerber file GM1 shows board outline (milling line)
 - Table 1 shows Layer stack details

Additional notes:

A1. Finished board thickness = 68.5mils +/- 10%; measured over top/bottom copper and solder mask

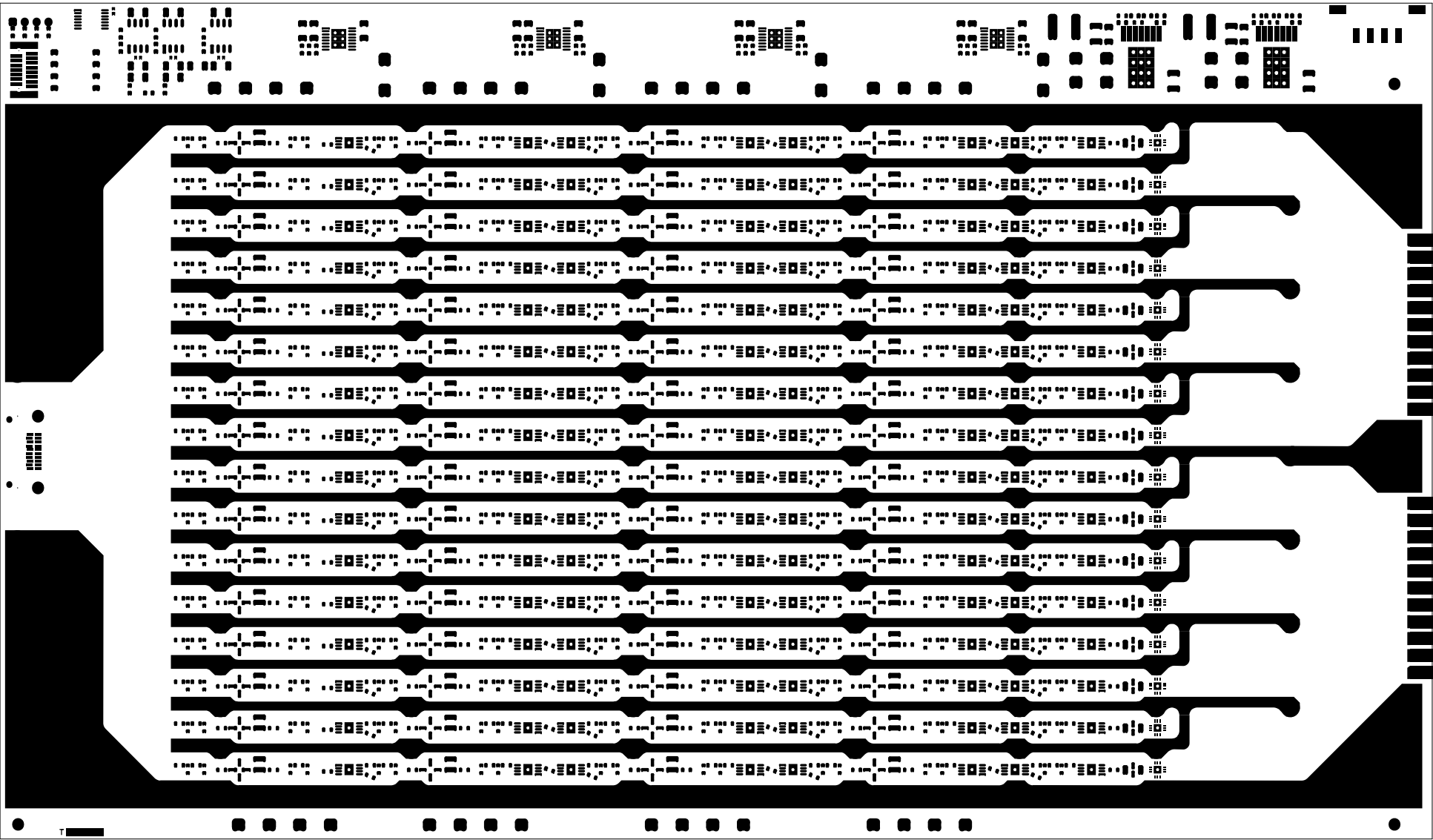
A1. Blind vias are from L1 to L2 and L5 to L6

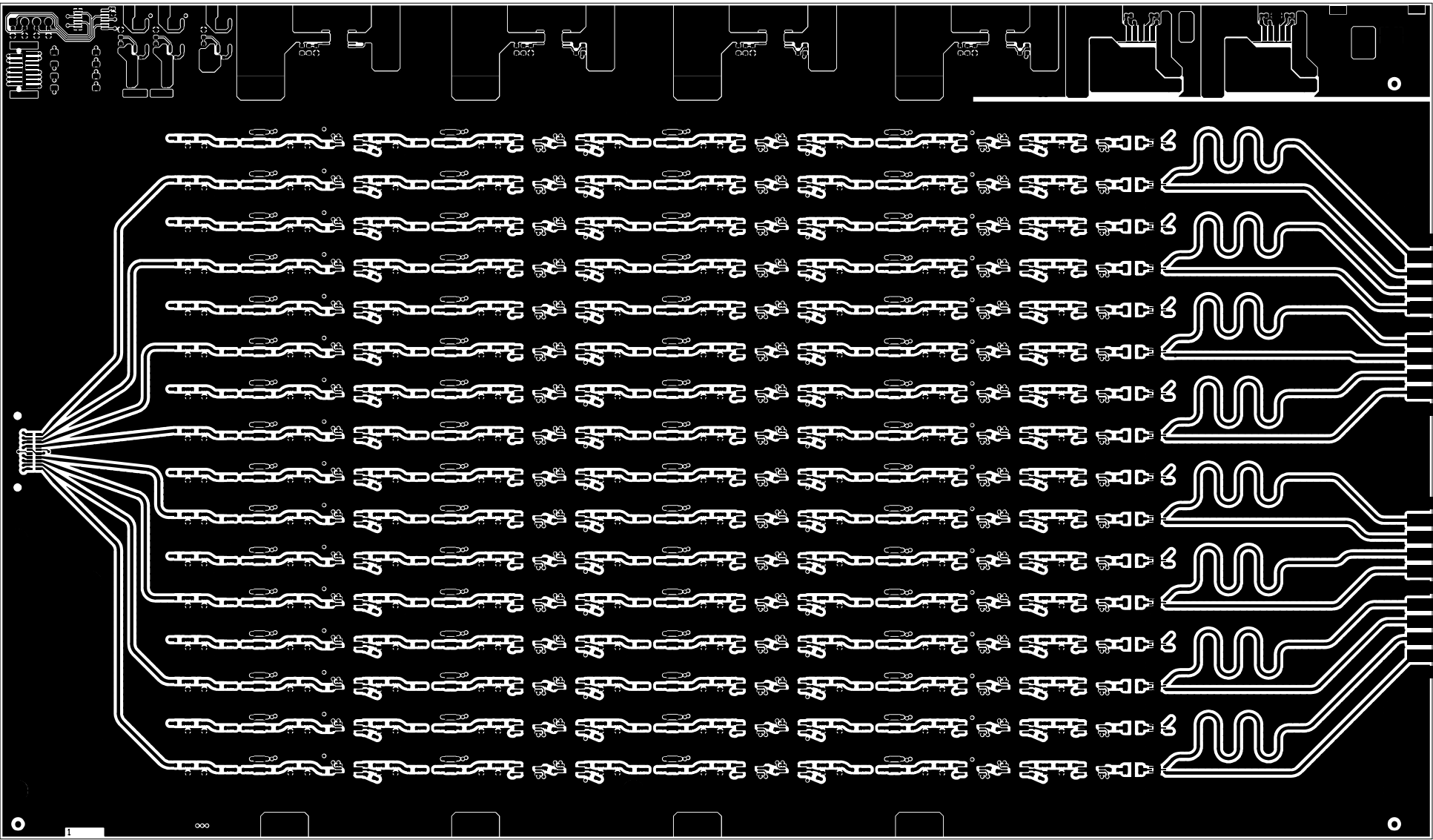
Table 1: Layer Stack Details for IDL_15_38 Rev.A (Imperial Units)

Layer	Name	Material	Thickness	Constant	Board Layer Stack
1	Top Overlay				
2	Top Solder	Solder Resist	1.50mil	3.5	
3	Top Layer - SIG1	Copper	1.38mil		
4	Dielectric 1	FR4	15.00mil	4.65	
5	Layer 2 - GND1	Copper	0.70mil		
6	Dielectric 2	FR4	9.00mil	4.65	
7	Layer 3 - SIG2	Copper	0.70mil		
8	Dielectric 3	FR4	12.00mil	4.65	
9	Layer 4 - PWR	Copper	0.70mil		
10	Dielectric 4	FR4	9.00mil	4.65	
11	Layer 5 - GND2	Copper	0.70mil		
12	Dielectric 5	FR4	15.00mil	4.65	
13	Bottom Layer - SIG3	Copper	1.38mil		
14	Bottom Solder	Solder Resist	1.50mil	3.5	
15	Bottom Overlay				

Table 2: NC Drill Details for IDL_15_38 Rev.A

Symbol	Hil Count	Finished Hole Size	Plated	Hole Type
□	2152	0.300mm (11.81mil)	PTH	Round
	2152 Total			





Designer: PO	Revision: .Version	File: IDL_15_38_A.PcbDoc	Sheet 1 of 1	Code: IDL_15_38
Drawn By: PO	Modif. Date: Date	Variant: [No Variations]	PCB	ID: XRM_MainAMP
Approved By: Gary S. Varner	Print Date: 3. nov 2015	Signature:	Size: A3 H	
Title: Top Layer- SIG1 (GTL)				University of Hawaii at Manoa High Energy Physics Group Instrumentation Development Laboratory

