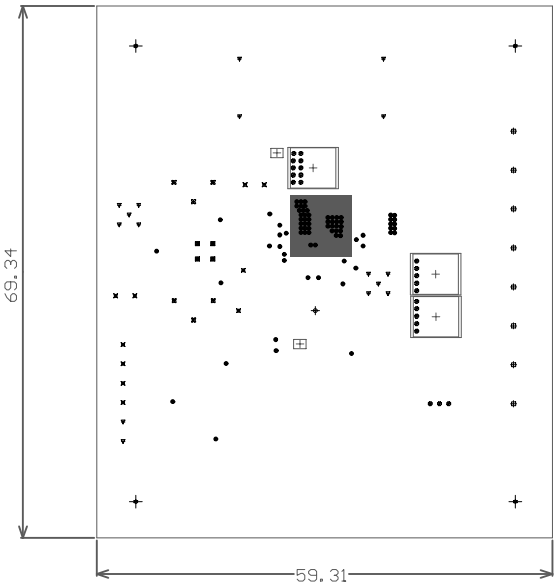
Symbol	Count	Hole Size	Plated	Hole Type	Drill Layer Pair	Via/Pad	Pad Shape	Template	Description	Hole Tolerance (+)	Hole Tolerance (-)
○	1	2.200mm (86.61mil)	PTH	Round	TopLayer - BottomLayer	Pad	Rounded	c300h220			
▣	4	0.850mm (33.47mil)	PTH	Round	TopLayer - BottomLayer	Pad	Rounded	c150h85			
✡	4	3.302mm (130.00mil)	NPTH	Round	TopLayer - BottomLayer	Pad	Rounded	c318hn330			
⊠	6	0.750mm (29.53mil)	PTH	Round	TopLayer - BottomLayer	Pad	Rounded	c140h75			
□	8	1.651mm (65.00mil)	PTH	Round	TopLayer - BottomLayer	Pad	Rounded	c305h165			
⊠	10	0.800mm (31.50mil)	PTH	Round	TopLayer - BottomLayer	Pad	Rounded	(Mixed)			
▽	16	0.900mm (35.43mil)	PTH	Round	TopLayer - BottomLayer	Pad	(Mixed)	(Mixed)			
✪	23	0.381mm (15.00mil)	PTH	Round	TopLayer - BottomLayer	Via	Rounded	v76h38m0mx0			
◎	78	0.254mm (10.00mil)	PTH	Round	TopLayer - BottomLayer	Via	Rounded	(Mixed)			
	150 Total										



Layer	Name	Material	Thickness	Constant	Board Layer Stack
	TopOverlay				
	TopSolder	Solder Resist	0.010mm	3.5	
1	TopLayer		0.070mm		
	Dielectric1	PP-006	0.239mm	4.1	
2	Signal Layer 1		0.070mm		
	Dielectric 3	Core-009	0.513mm	4.5	
3	Signal Layer 2		0.070mm		
	Dielectric 2	PP-006	0.239mm	4.1	
4	BottomLayer		0.070mm		
	BottomSolder	Solder Resist	0.010mm	3.5	
	BottomOverlay				

General Notes:
1. Boards are to be manufactured to IPC 6012.
2. All holes shall be plated through unless otherwise specified.
3. Copper: 4 layers, 2oz (70um) all layers.
4. Finish: Green soldermask both sides.
5. Silkscreen shall be legible and white in color.
6. Gold Plated
7. PCB total thickness should be less than 1.3mm