Design Rules Verification Report

Filename : D:\files\Altium files\Altium projects\Water Sprkling\PCB.PcbDoc

<div align="right">

Warnings 0
Rule Violations 0

</div>

Warnings

Total	0

Rule Violations

Clearance Constraint (Gap=0.2mm) (All),(All)	0
Short-Circuit Constraint (Allowed=No) (All),(All)	0
Un-Routed Net Constraint ((All))	0
Modified Polygon (Allow modified: No), (Allow shelved: No)	0
Width Constraint (Min=0.1mm) (Max=1.5mm) (Preferred=1mm) (InNet('NetF_2'))	0
Width Constraint (Min=0.1mm) (Max=1.5mm) (Preferred=0.8mm) (All)	0
Width Constraint (Min=0.254mm) (Max=1mm) (Preferred=1mm) (InNet('NetF_2'))	0
Width Constraint (Min=0.1mm) (Max=1.5mm) (Preferred=1mm) (InNet('LINE_1'))	0
Width Constraint (Min=0.1mm) (Max=1.5mm) (Preferred=1mm) (InNet('LINE_2'))	0
Width Constraint (Min=0.254mm) (Max=1mm) (Preferred=1mm) (InNet('LINE_1'))	0
Power Plane Connect Rule(Relief Connect)(Expansion=0.508mm) (Conductor Width=0.254mm) (Air Gap=0.254mm)	0
Hole Size Constraint (Min=0.025mm) (Max=20mm) (All)	0
Hole To Hole Clearance (Gap=0.254mm) (All),(All)	0
Minimum Solder Mask Sliver (Gap=0.254mm) (All),(All)	0
Silk To Solder Mask (Clearance=0.01mm) (IsPad),(All)	0
Silk to Silk (Clearance=0.1mm) (All),(All)	0
Net Antennae (Tolerance=0mm) (All)	0
Height Constraint (Min=0mm) (Max=100mm) (Prefered=60mm) (All)	0
Total	0