
1. Hole locations are indicated in separate .drl file, included with the gerber package. That file takes precedence over this drawing.
2. Board outline indicated in separate Edge.Cuts file, included with the gerber package. That file takes precedence over this drawing.
3. Vendor to plate holes and exposed copper pads
4. Board Thickness: 0.062in (1.57mm) or 1.6mm
5. Material: FR-4
6. PCB Color: Green
7. SilkScreen: White
8. Surface Finish: HASL - RoHS
9. Copper Weight: 1 oz

