



- 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.) Total board height: 1.6mm +/- 10 %

Total: 1530  $\mu\text{m}$

1. Material: FR-4 (TG-150)
2. Soldermask color: Black/Blue
3. Silkscreen color: White
4. Surface Finish: ENIG
5. Impedance Control: Yes
6. VIAs: Tented with soldermask
7. Stackup: See stackup table below for layer specific info

Id: 1/1