Layer (Material) Type	Layer (File) Name	Thickness (mm)	Dielectric Constant	Notes
Silkscreen	F.Silk	0.00762		
Soldermask	F.Mask	0.0127	3.8	
Copper	F.Cu	0.035		
FR4 (Core)	-----	1.6		
Copper	B.Cu	0.035		
Soldermask	B.Mask	0.0127	3.8	
Silkscreen	B.Silk	0.00762		

Notes:

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

PCB Edge