

## PCB Fabrication

<b>Layers</b>	4
Arrays?	No
<b>X Dimension</b>	0.646"
<b>Y Dimension</b>	0.51"
Route and Retain	No
Scoring	No
<b>Material</b>	FR4
<b>Board Thickness</b>	0.031"
<b>Surface Finish</b>	HASL
Gold Fingers	No
<b>Outer Layer Finish Copper</b>	1 oz
<b>Inner Copper</b>	1 oz
Counter Bore	No
Counter Sink	No

<b>Solder Mask</b>	Yes
<b>Solder Mask Sides</b>	Top and Bottom
<b>Solder Mask Color</b>	Green
<b>Solder Mask Type</b>	LPI
<b>Solder Mask Finish</b>	Standard

<b>Silk Screen</b>	None
<b>Silk Screen Sides</b>	-
<b>Silk Screen Color</b>	-
Internal Slots and/or Internal cutouts	No
Edge Plating	No
Controlled Impedance	No
Controlled Dielectric	No

<b>Layers</b>	<b>4</b>
---------------	----------

## Layer Order

Layer	Gerber File
Top soldermask	.stc
Top signal	.cmp
Ground Layer	.ly2
Power Layer	.ly15
Bottom signal	.sol
Bottom soldermask	.sts

## Gerber Files

Extension	DESCRIPTION
.cmp	Top side copper
.cmp_ref	Top side reference designators
.drd	Excellon drill description
.dri	Excellon drill tool description
.drl	Drill rack data
.g1	Board outline
.gpi	Gerber photoplotter information
.ly2	Inner layer (2 of 6)
.ly15	inner layer (5 of 6)
.sol	Bottom side copper
.sol_ref	Bottom side reference designators
.stc	Top side solder mask
.sts	Bottom side solder mask