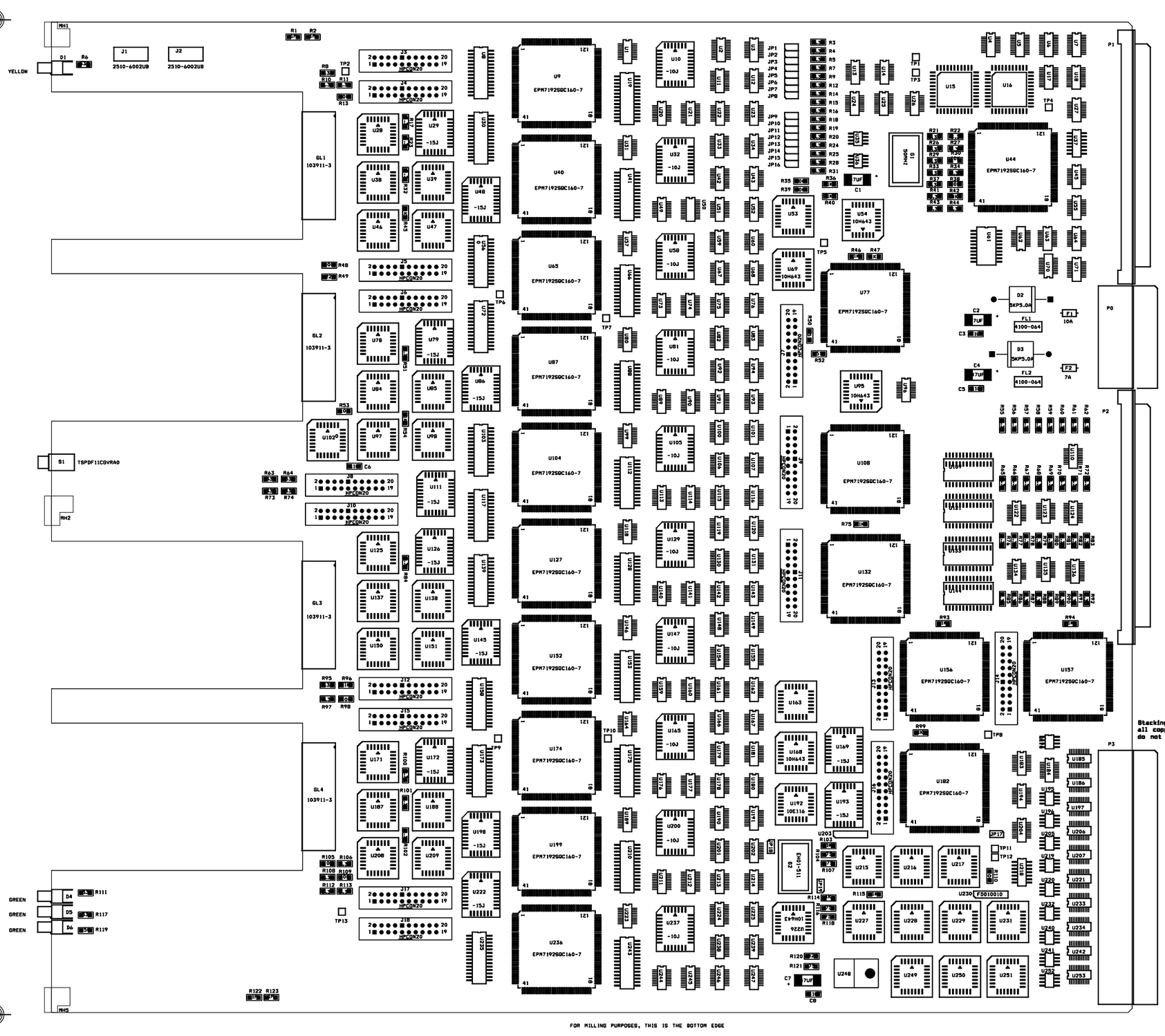
FOR MILLING PURPOSES, THIS IS THE TOP EDGE



Stacking stripes on all copper layers: do not remove:

FOR MILLING PURPOSES, THIS IS THE BOTTOM EDGE

Materials: FR-4 with Tg of 140 or greater.
Maximum Panel size of ☐12" ■18" ☐24"
Horizontal etch limits: No greater than -.001" in 1/2 oz. Cu layers.
No greater than -.002" in 1 oz. Cu layers.
No greater than -.004" in 2 oz. Cu layers.
Copper plating on PTH walls: Minimum 1 mil plating in the hole.
Drill Hole Tolerances: 0.050 in. diameter or less: +/- 0.003 in.
Drill Hole Tolerances: 0.050 in. diameter or more: +/- 0.007 in.
No drill hole shall breakout from any pad. Minimum annular copper ring radius: 0.002 in.
SMOBC
LPI Soldermask
Silkscreen color: White
HASL
Testing Required: ☐ None
■ Flying Probe
☐ Clamshell

Technical Contact: James Franzen
Voice phone: 630-840-2709
Fax: 630-840-8208
Email: franzen@fnal.gov

Gerber Lists:

■ TAP (Drill Tape)	■ APP (Aperture List)
☐ FAB (Fabrication Drwg)	■ DRD (Drill Drwg)
■ DTB (Drill Tape Summary)	■ DTD (OrCAD Extended Gerber)
■ ABT (Top Assembly)	■ ABB (Bottom Assembly)
■ SPT (Solder Paste Top)	■ SPB (Solder Paste Bottom)
■ SST (Silkscreen Top)	■ SSB (Silkscreen Bottom)
■ SMT (Solder Mask Top)	■ SMB (Solder Mask Bottom)
■ TOP (Top Signal)	■ BOT (Bottom Signal)
■ GND (Ground Plane)	■ PWR (Power Plane)
■ IN1 (Inner Signal 1)	☐ IN5 (Inner Signal 5)
■ IN2 (Inner Signal 2)	☐ IN6 (Inner Signal 6)
■ IN3 (Inner Signal 3)	■ PW3 (VTT Plane)
■ IN4 (Inner Signal 4)	■ PW2 (VEE Plane)

*Regarding the controlled impedance of this board:
1)The finished thickness of this board shall be .098" +/- .008".
2)The target impedance for all signal layers is as high as possible without exceeding 100 ohms.
3)The stakup of this board shall be such that the impedance of any signal layer shall have no greater deviation than 10% from the median impedance of all signal layers.

LAYER STACKUP:
1) Top Signal (*controlled impedance microstrip)
2) VCC Plane
3) IN1 Signal (*controlled impedance dual stripline)
4) IN2 Signal (*controlled impedance dual stripline)
5) GND Plane
6) VEE Plane
7) IN3 Signal (*controlled impedance dual stripline)
8) IN4 Signal (*controlled impedance dual stripline)
9) VTT Plane
10) Bottom Signal (*controlled impedance microstrip)

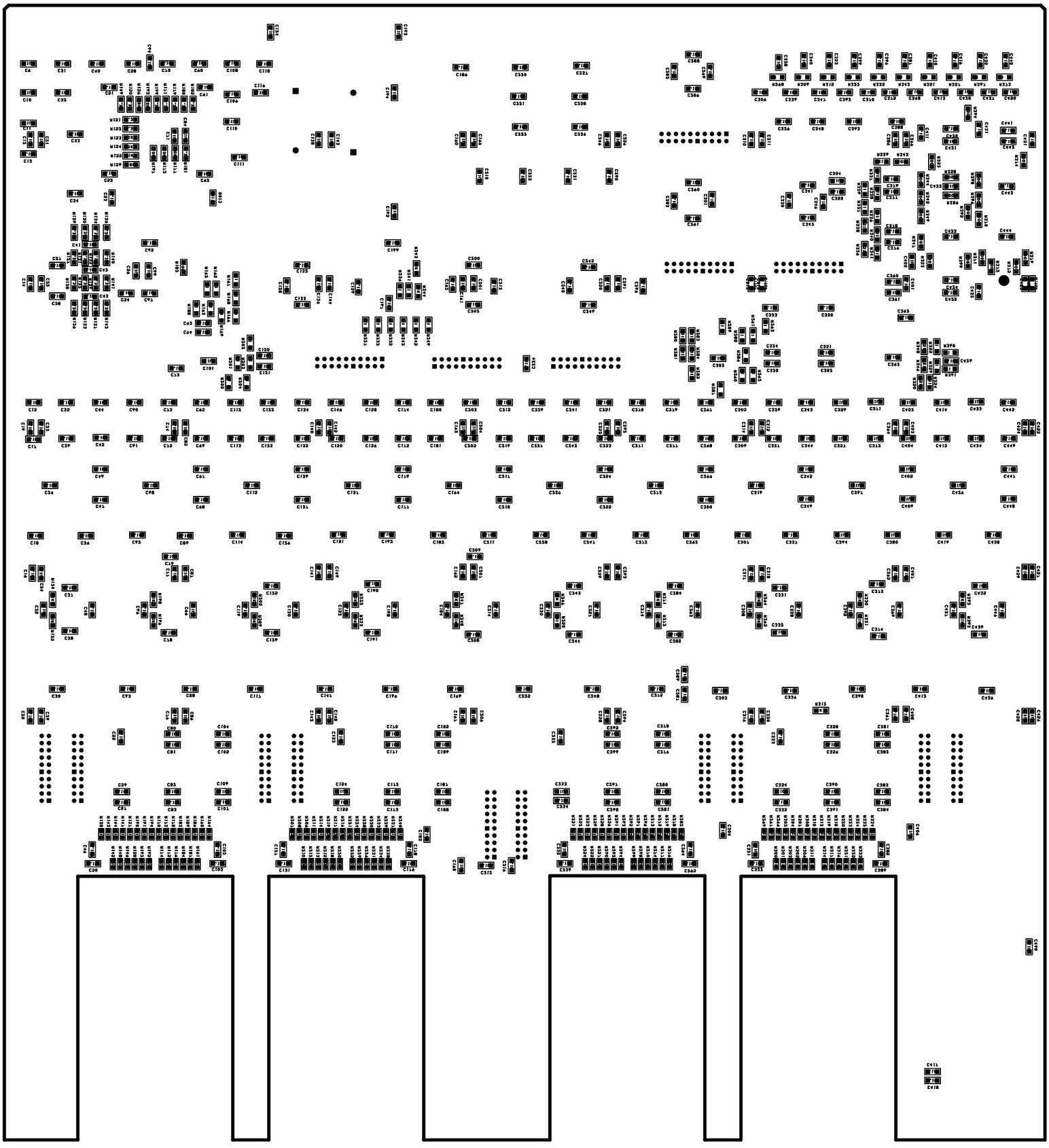
Finished board thickness is 0.093" +/-.008".
Mill solder side (bottom) of board along both top and bottom edges.
Mill width is 0.098" +0.000" -0.008" in from the top and bottom edges.
Finished board thickness of milled edge area is to be 0.063" +/-.008".

Two corners are chamfered, refer to dimensional information located on the Drill Drawing (.DRD) Gerber.

Unless noted otherwise, dimensional tolerances are +/- .003"

GERBER EXTENSIONS:

.SPT Top Solderpaste
.SST Top Silk Screen
.SMT Top Soldermask
.TOP Top Signal
.PWR VCC Plane
.IN1 Inner Signal 1
.IN2 Inner Signal 2
.GND Ground Plane
.PW2 VEE Plane
.IN3 Inner Signal 3
.IN4 Inner Signal 4
.PW2 VTT Plane
.BOT Bottom Signal
.SMB Bottom Soldermask
.SSB Bottom Silk Screen
.SPB Bottom Solderpaste
.ABT Assembly Top
.ABB Assembly Bottom
.DRD Drill Drawing
.DRL Drill Holes

.SPT Top Solderpaste
.ABT Assembly Top

.SPB Bottom Solderpaste
.ASB Assembly Bottom