

# **Super CDMS**

## **Detector Control and Readout Card**

---

**Printed Circuit Board  
Fabrication Specification**

**Design Rev. F.**

Updated: 2021-06-30

---

Sten Hansen (Design Engineer)  
Johnny Green (Procurement Specialist)  
Jamieson Olsen

---



U.S. DEPARTMENT OF  
**ENERGY**

Office of  
Science

## Contact Information

---

Johnny Green  
Fabrication Specialist  
Fermilab  
Mail Stop 222  
Kirk Road and Pine Street  
Batavia, Illinois, USA 60510-0500  
Phone: 630-840-3392  
e-mail: [jbgreen@fnal.gov](mailto:jbgreen@fnal.gov)

## Quantity and Delivery

---

Quantity = 40  
Delivery = Quote 5 and 10 day turn

## General Board Information

---

Board Dimensions = 11.0" x 6.0"  
Thickness = 0.093"  
Board Material Substrate = High Temperature FR4 or equivalent (subject to approval)  
Layer Count = 6 (4 signal and 2 plane)  
Finish = SMOBC  
Plating = ENIG  
Lead Free / RoHS = YES  
Copper Weight = 1 oz (outer layers), ½ oz (inner layers)  
Gold Contacts = NO  
Process Rails = YES, vendor to add, see below.  
Minimum Trace = 0.005"  
Minimum Space = 0.005"  
Solder Mask = YES, Top and Bottom, LPI, Green  
Solder Mask Tenting = YES  
Silkscreen Legend = YES, Top and Bottom, White  
SMT Pad Count = 5132 (Top), 1413 (Bottom)  
Through Hole Count = 3502  
Through Hole Drill Sizes = 20  
Through Hole Smallest = 0.008"  
Through Hole Largest = 0.128"  
Through Hole Tolerance = +/- 0.003"  
Blind Vias = NO  
Buried Vias = NO

Controlled Impedance = NO

Electrical Test = YES, netlist test

General Fabrication Specification = IPC-A-600E

## **Board Stackup / Fabrication Files**

---

<https://supercdms-docdb.fnal.gov/cgi-bin/sso/ShowDocument?docid=4394>

DCRC\_RevF\_Gerbers.zip contains:

### **Gerber Files, 274X**

Layer 1 = \*.TOP

Layer 2 = \*.IN1

Layer 3 = \*.GND (plane)

Layer 4 = \*.PWR (plane)

Layer 5 = \*.IN2

Layer 6 = \*.BOT

Solder Mask Top = \*.SMT

Solder Mask Bottom = \*.SMB

Silkscreen Top = \*.SST

Silkscreen Bottom = \*.SSB

Drill File = \*.DRD

Assembly Top Drawing = \*.ASB

Assembly Bottom Drawing = \*.ASB

Solder Paste Top = \*.SPT

Solder Paste Bottom = \*.SPB

### **Other Files**

Gerbtool Project File = \*.GTD

NC File = thruhole.tap

Pick and Place File = INSERT.TXT

OrCAD Project File = \*.DSN

Netlist = \*.MNL

### Addition of Process Rails

---

Vendor to add 0.5" process rails along long edges of the PCB.

Break away mechanism should be scoring or "mouse bites".

Vendor to send modified Gerber files to Fermilab for approval.

### Copper Thieving

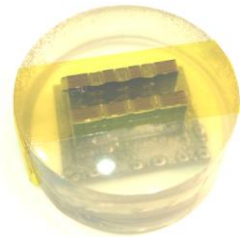
---

Allowed, subject to Fermilab approval.

### Quality Control

---

Vendor to supply test coupons to evaluate hole plating.



Vendor to conduct NETLIST electrical test and clearly mark each PCB with PASS indicator.