Fermi National Accelerator Laboratory

Super CDMS
Detector Control and Readout Card

Assembly Specification

Design Rev. F.

Updated: 2021-07-01

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

U.S. DEPARTMENT OF ENERGY | Office of Science

Managed by Fermi Research Alliance, LLC for the U.S. Department of Energy Office of Science

www.fnal.gov

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

Documentation Location

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

username = review
password = snowlab2015

General Information

The DCRC stands for "Detector Control and Readout Card". The current revision of this design is F. The board is a mixed signal design incorporating fairly high-speed digital backend electronics and sensitive low noise analog front-end electronic components.

PCB Information

The DCRC measures 11.0" x 6.0" x 0.093" FR4, 6-layer printed circuit board with approximately 1885 components of which 250 are unique. Most of the components on the board are surface mount consisting BGA (Cyclone 4 484 BGA), SOT23, MSOP, TSSOP, SQFP, with 20, 25 mils and 50 mil pitch packages. There are 0402 and 0603 discreet components and some thru-hole components. The PCB is not impedance controlled. Surface is SMOBC with ENIG plating. All PCBs will be 100% Netlist tested by the PCB vendor. Each PCB will be 1UP and have 0.500" process rails along the long edges.

Quantity and Delivery Schedule

42 assemblies total.
2 assemblies for first piece approval will be assembled in two weeks.
First piece inspection approval to be 1 week or less, then the approval will be given for the remainder.
After approval one unit will be returned to be used as the "golden sample" for the remainder of the build.
The remaining 40 assemblies after the first piece approval will be delivered after 4 weeks.

Parts

The BOM (Excel Spreadsheet) will contain all known components to be assembled including those to be supplied by Fermilab. Fermilab to supply PCBs and some parts as indicated in the BOM. The vendor will supply all remaining parts, subject to Fermilab approval.

Additions, Deletions, and Substitutions initiated by Fermilab before Parts Approval

Additions, deletions, and substitutions to the parts list can occur for each assembly phase will be handled as follows: 1) Fermilab must approve all parts changes; 2) additional parts will be assigned a new Fermilab assigned number (F/N).

Additions, Deletions, and Substitutions initiated by Fermilab after Parts Approval

Additions, deletions, and substitutions to the parts list can occur for each assembly phase and will be handled as follows: 1) vendor will notify Fermilab of parts shortages and Fermilab will procure them; 2) additional parts will be assigned a new Fermilab assigned number (F/N).

There are Do Not Install components (DNI). See BOM and pick-n-place file for details.
Please review the assembly drawings for other pertinent information.

Applicable Standards

Assembly must be ROHS compliant.
Vendor is required to fully conform to ANSI/IPC-A-610. Class 2.

Serial Numbers

All boards shall be serialized, and these serial numbers referenced by inspection documentation.

Board Warpage

Bare board warpage is specified at less than 0.005 inches per linear inch. Board warpage after assembly must be less than 0.010 inches per linear inch.

Assembled Board Inspection and Rework

Vendor is required to provide documentation of a thorough visual inspection using AOI. Vendor is required to provide documentation of X-Ray inspection of the BGA to insure proper solder reflow. BGA X-Ray images must reference the board serial number. Any misplaced, rotated, or parts with the wrong values should be identified and repaired by whatever inspection means used as a complementary inspection by vendor.

Cleaning and Packaging

Final assembled product must be free of any flux/chemicals. “no clean” flux shall not be used. We prefer a deionized water cleaning no matter which solder/flux is used. Vendor shall remove the process rails from the long edges of the PCB. Finished Boards to be packaged in ESD bags with appropriate protection to avoid shipping damage.

Fabrication Files

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

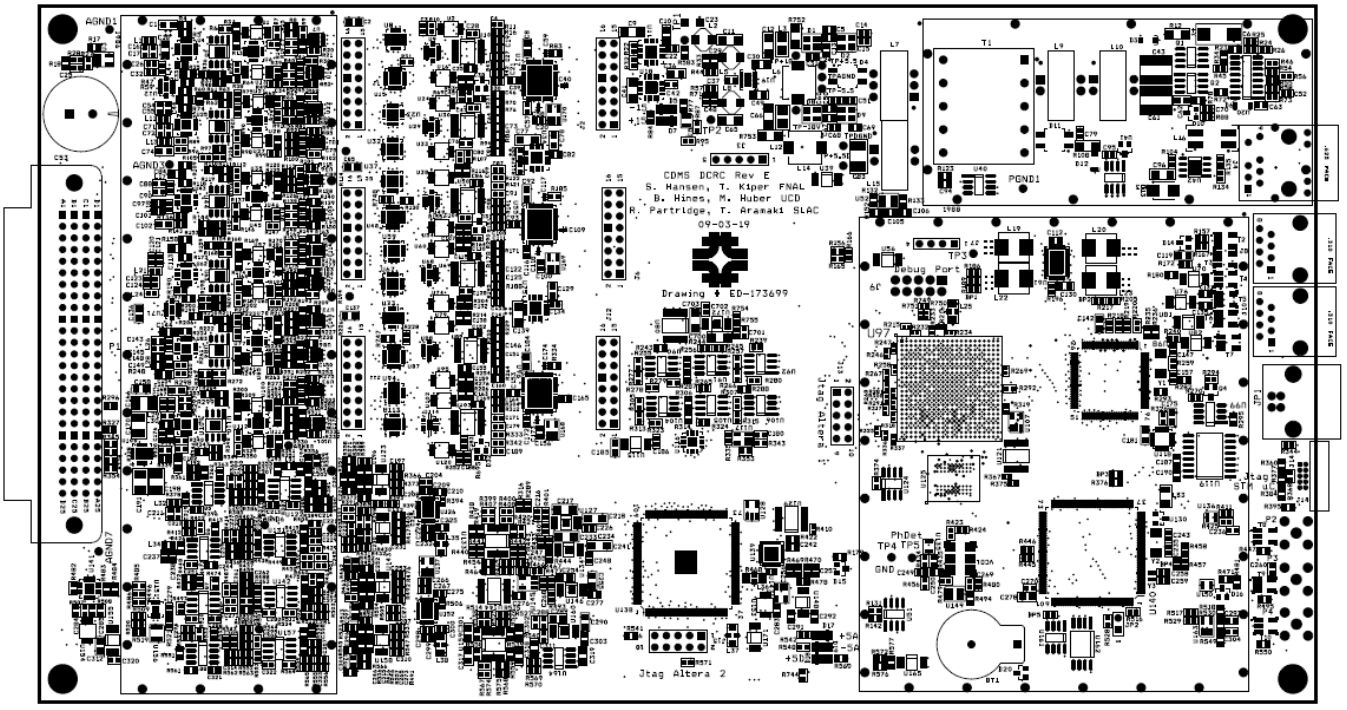
Other Files

Gerbtool Project File = *.GTD
NC File = thruhole.tap
Pick and Place File = INSERT.TXT

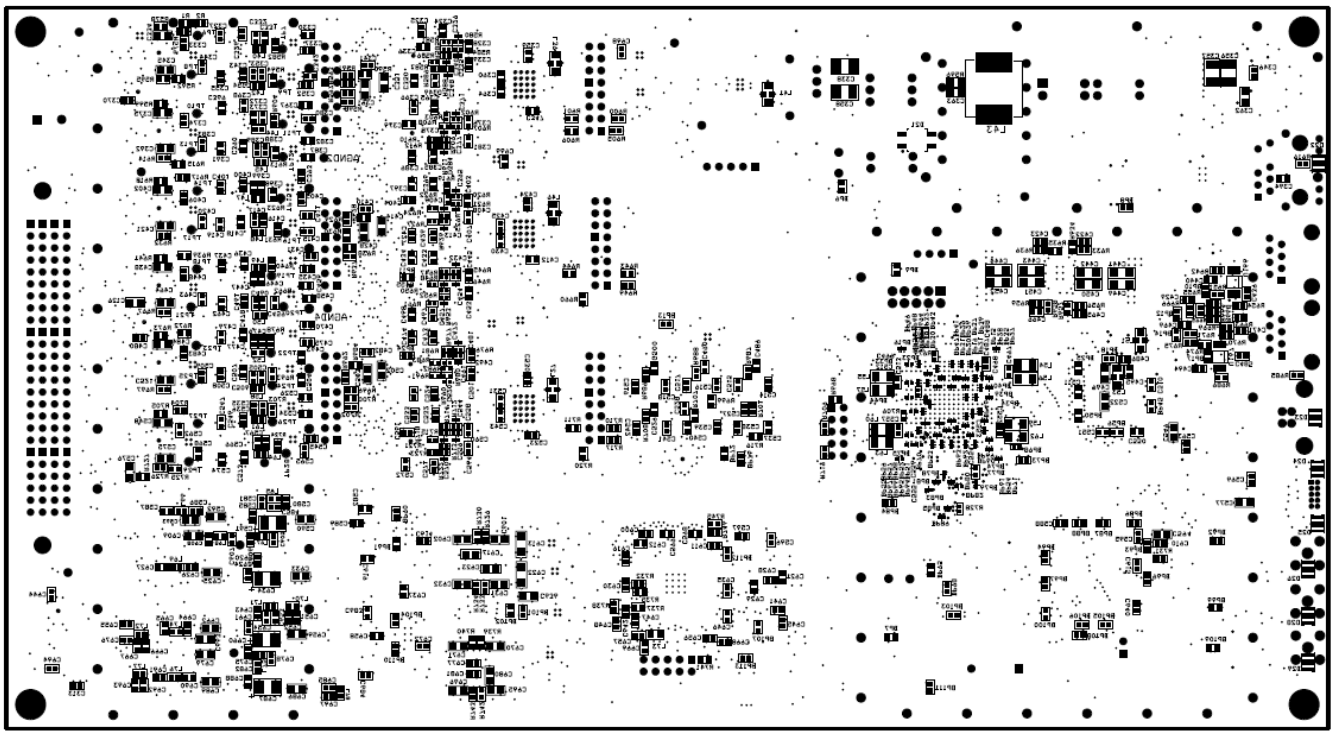
Top and Bottom PCB Drawings

The following images are provided for reference only. Refer to the Gerber files for accurate high resolution drawings.

DCRC Top Solder Mask-Silkscreen



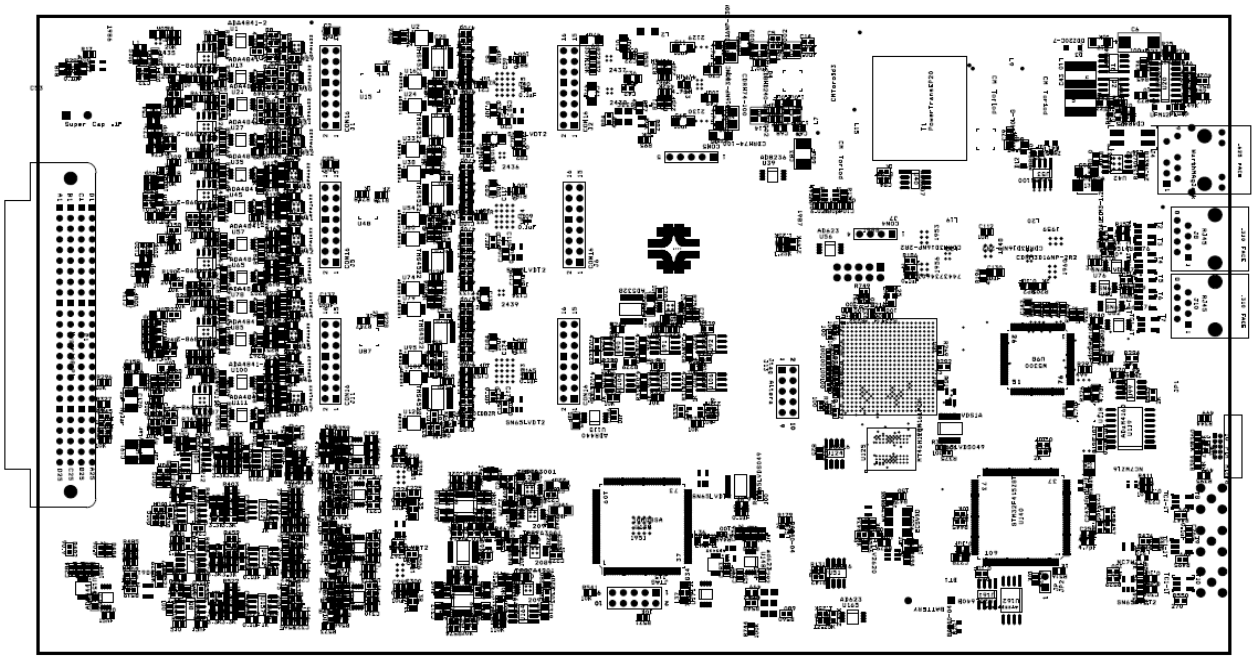
DCRC Bottom Solder Mask-Silkscreen



Assembly Drawings

The following images are provided for reference only. Refer to the Gerber files for accurate high resolution drawings.

Assembly Top Drawing



Assembly Bottom Drawing