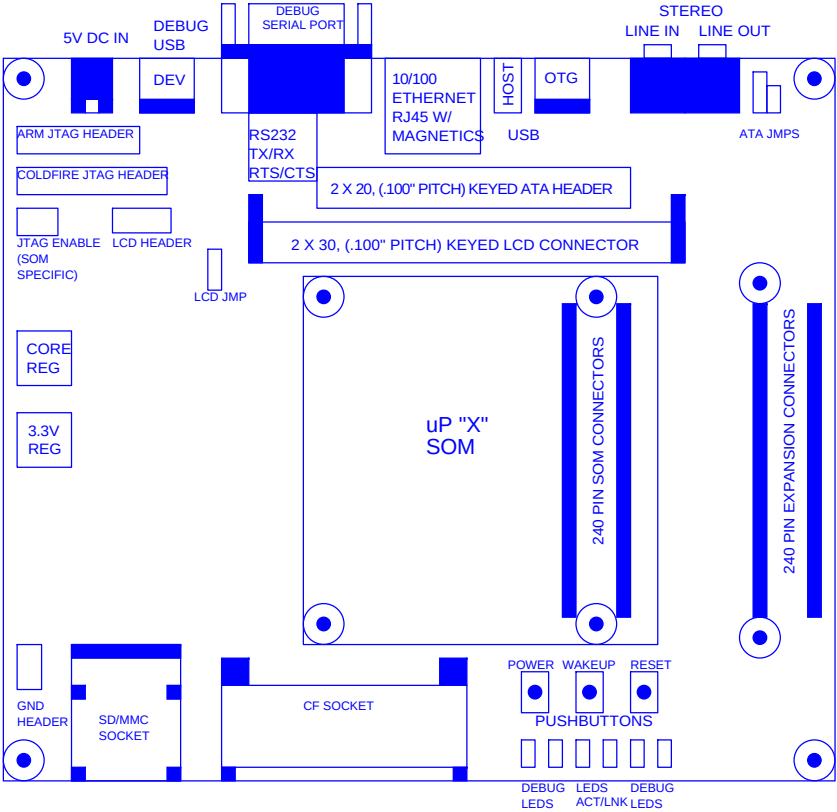


PROJECT:	SDK2
PART NUMBER:	1005441
ASSEMBLY NAME:	SDK2 BASEBOARD
SCHEMATICS:	KTL/ECR
TABLE OF CONTENTS	
PAGE	DESCRIPTION
1	TITLE PAGE AND BLOCK DIAGRAM
2	MODULE BUS
3	ETHERNET
4	AUDIO
5	CF CARD
6	RS232 UART
7	LCD
8	POWER
9	JTAG
10	EXPANSION BOARDS
11	USB
12	SD/MMC/ATA
13	ECO LIST



IMPORTANT NOTES ABOUT THIS SCHEMATIC

DESIGN NOTE: Example text for the design note to show the note inside the colored box.

1) DESIGN NOTES in grey are information notes.

DESIGN NOTE: Example text for the design note to show the note inside the colored box.

2) DESIGN NOTES in yellow are notes of caution.

DESIGN NOTE: Example text for the design note to show the note inside the colored box.

3) DESIGN NOTES in red are critical, and must be understood and followed.

IMPORTANT NOTICE:

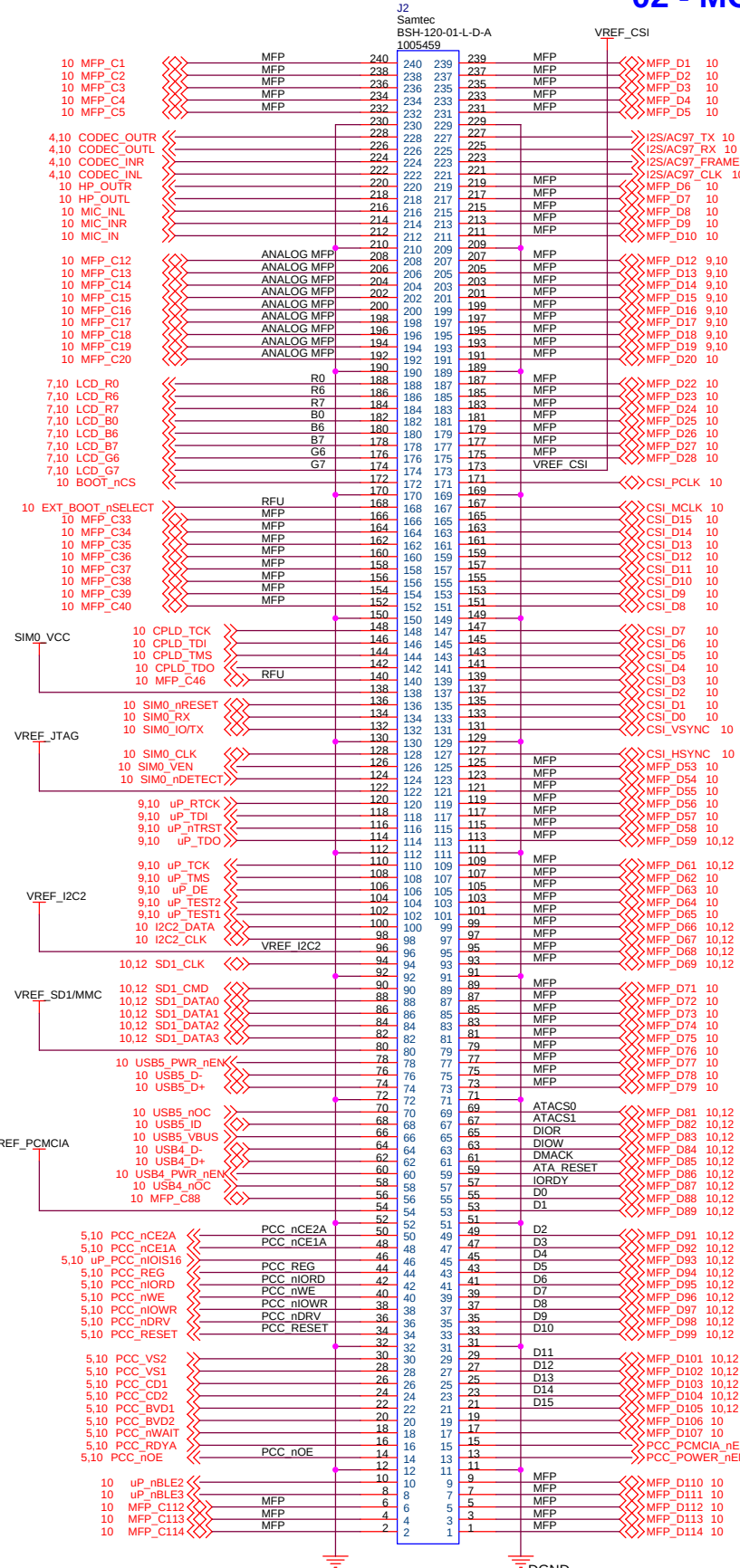
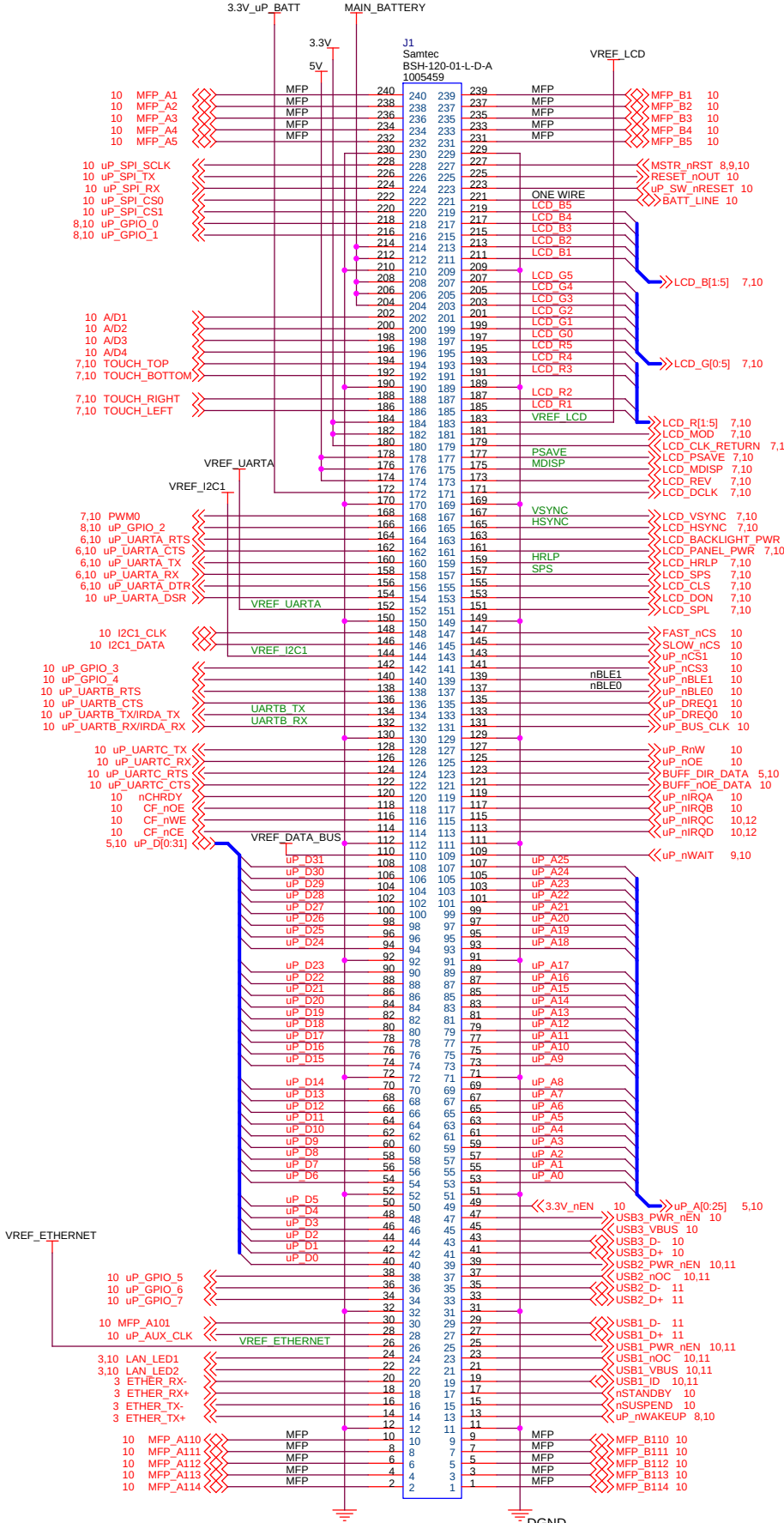
1. THIS DOCUMENT MAY NOT REFLECT THE MOST RECENT CHANGES IN BOARD DEVELOPMENT AND DEBUG. ANY DEVELOPER INTENDING TO USE THIS SCHEMATIC AS A REFERNCE SHOULD CONTACT THEIR THE LPD WEBSITE (WWW.LOGICPD.COM), REGIONAL SALES OFFICE, OR PROGRAM MANAGER FOR SCHEMATIC UPDATES, DESIGN RECOMMENDATIONS AND PCB LAYOUT GUIDELINES. LPD ALSO RECOMMENDS A DESIGN REVIEW OF BOTH THE SCHEMATIC DIAGRAM AND PCB LAYOUT BEFORE CONSIDERING PRODUCTION.

2. LPD RESERVES THE RIGHT TO CHANGE DESIGNS OR SPECIFICATIONS WITHOUT NOTICE. CUSTOMERS ARE ADVISED TO OBTAIN THE LATEST VERSIONS OF PRODUCT SPECIFICATIONS, WHICH SHOULD BE CONSIDERED IN EVALUATING A PRODUCT'S APPROPRIATENESS FOR A PARTICULAR USE.

3. LPD MAKES NO WARRANTIES, EXPRESSED OR IMPLIED, FOR MERCHANTABILITY OR FITNESS FOR A PARTICULAR APPLICATION. IN NO EVENT SHALL LPD BE LIABLE FOR ANY INDIRECT, SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES AS A RESULT OF THE PERFORMANCE, OR FAILURE TO PEFORM, OF ANY LPD PRODUCT OR DOCUMENTATION.

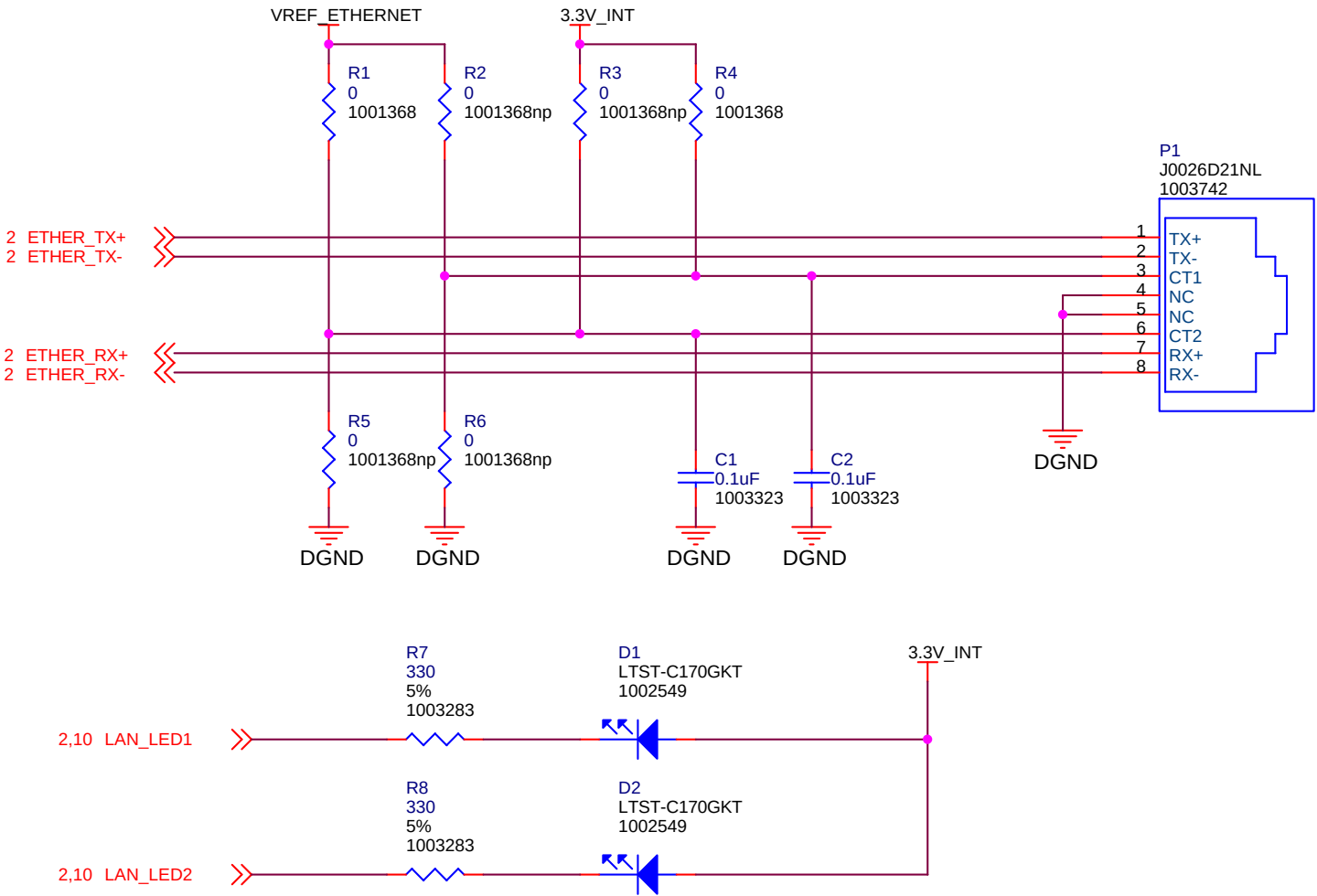
4. © 2007 LOGIC PRODUCT DEVELOPMENT. ALL RIGHTS RESERVED. LPD, THE LPD LOGO, COMBINATIONS THEREOF, ARE TRADEMARKS OF LOGIC PRODUCT DEVELOPMENT. OTHER NAMES USED IN THIS PUBLICATION ARE FOR IDENTIFICATION PURPOSES ONLY AND MAY BE TRADEMARKS OF THEIR RESPECTIVE COMPANIES.

02 - MODULE BUS

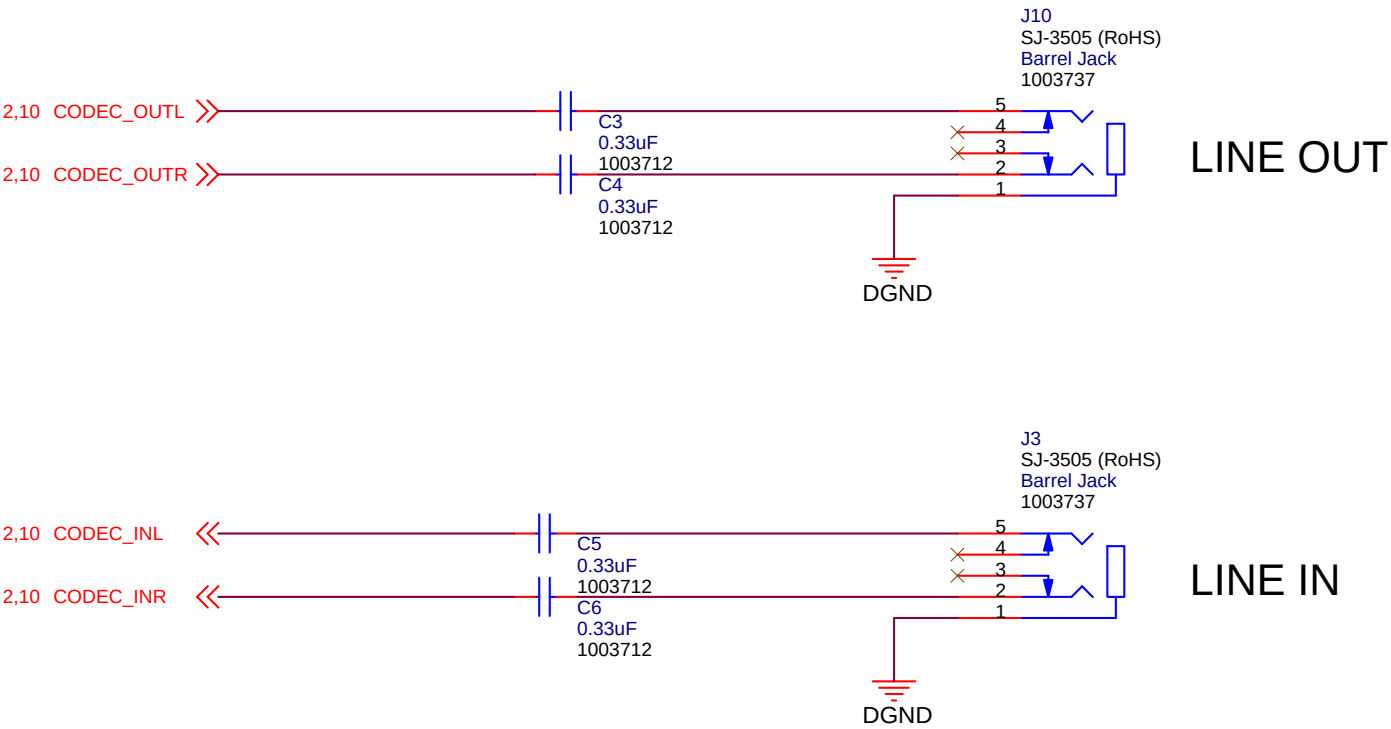


POPULATION CHART BY ETHERNET CHIP ON MODULE		
LAN9117	R1, R4	ALL OTHER NO POPULATED
DP83848 LAN9211	R1, R2	ALL OTHER NO POPULATED

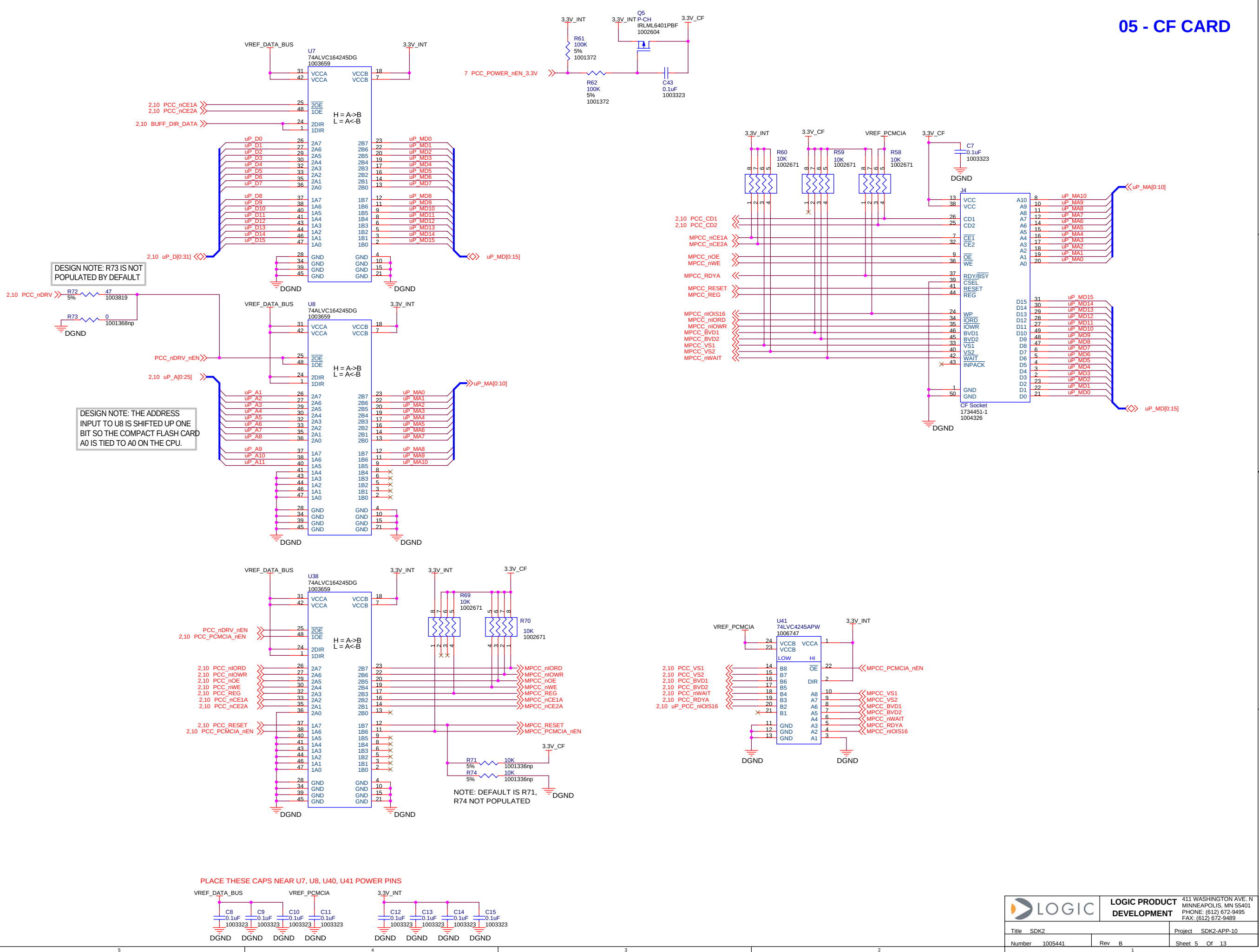
DESIGN NOTE: The Ethernet signals are differential pairs with a differential impedance of 100 Ohms. Each pair must be length matched and have a target impedance of 100 Ohms +/- 10%.



04 - AUDIO

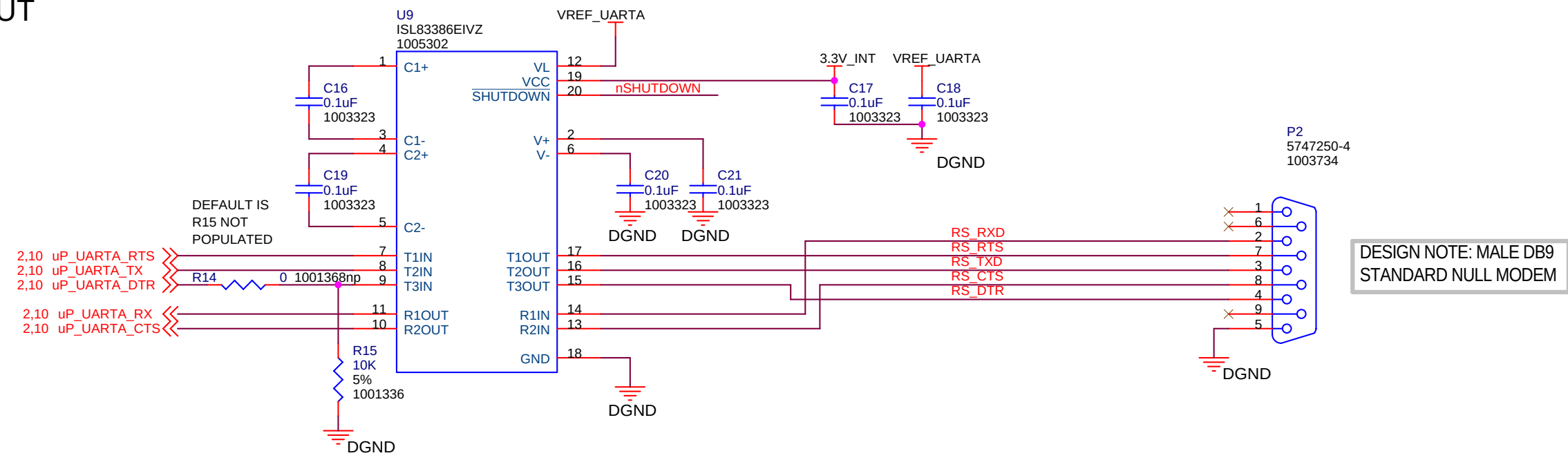


05 - CF CARD



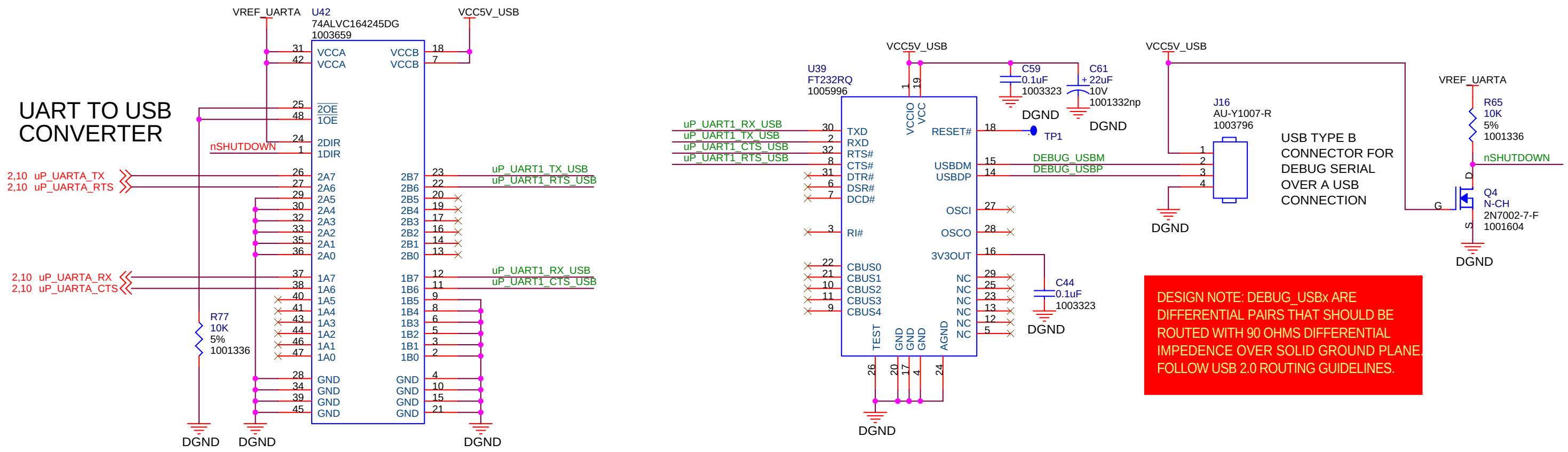
06 - RS232 UART

SERIAL OUTPUT



DESIGN NOTE: MALE DB9
STANDARD NULL MODEM

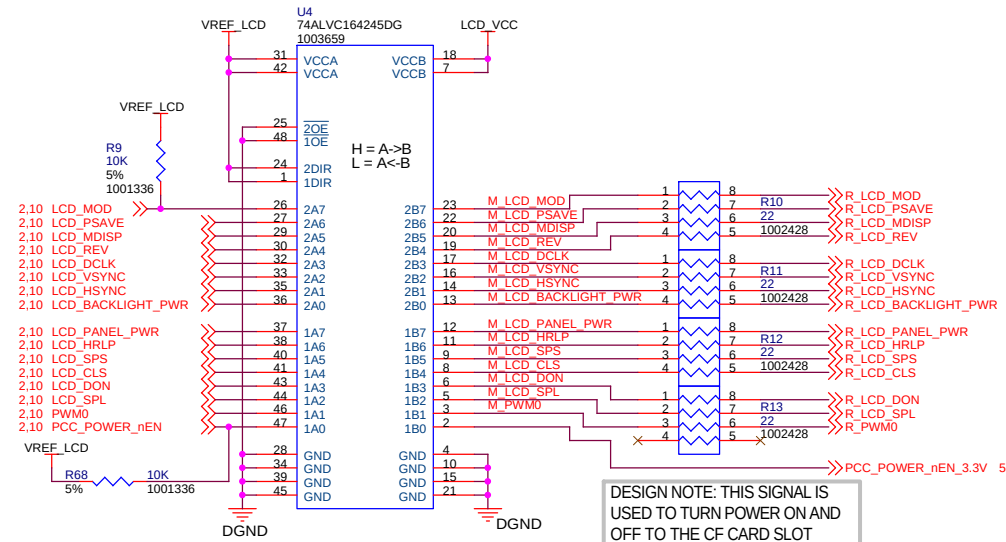
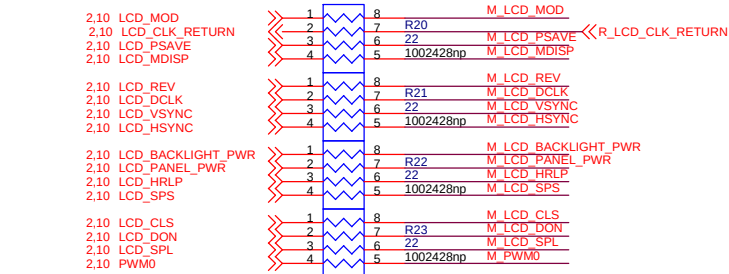
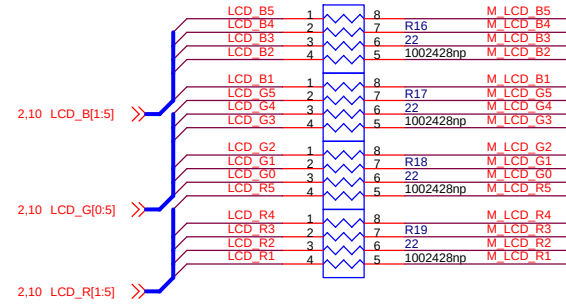
UART TO USB CONVERTER



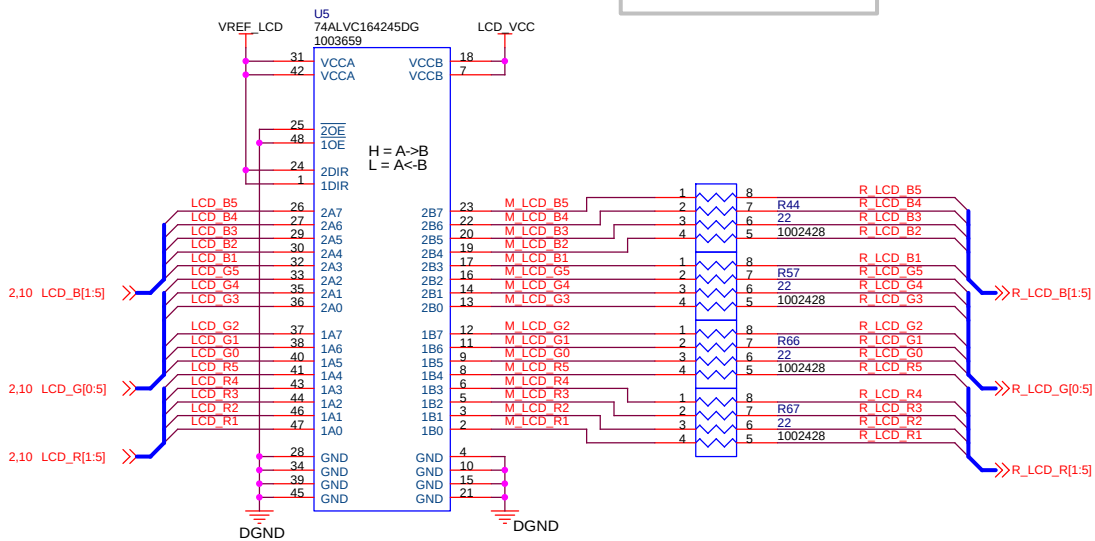
USB TYPE B
CONNECTOR FOR
DEBUG SERIAL
OVER A USB
CONNECTION

DESIGN NOTE: DEBUG_USBx ARE
DIFFERENTIAL PAIRS THAT SHOULD BE
ROUTED WITH 90 OHMS DIFFERENTIAL
IMPEDANCE OVER SOLID GROUND PLANE.
FOLLOW USB 2.0 ROUTING GUIDELINES.

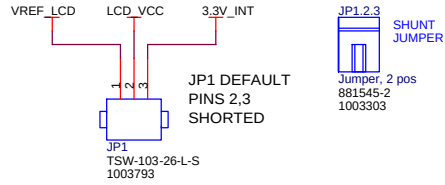
DESIGN NOTE: DEFAULT IS
RPACKS R16-R23 NOT POPULATED.



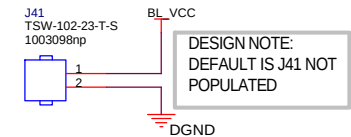
DESIGN NOTE: THIS SIGNAL IS
USED TO TURN POWER ON AND
OFF TO THE CF CARD SLOT



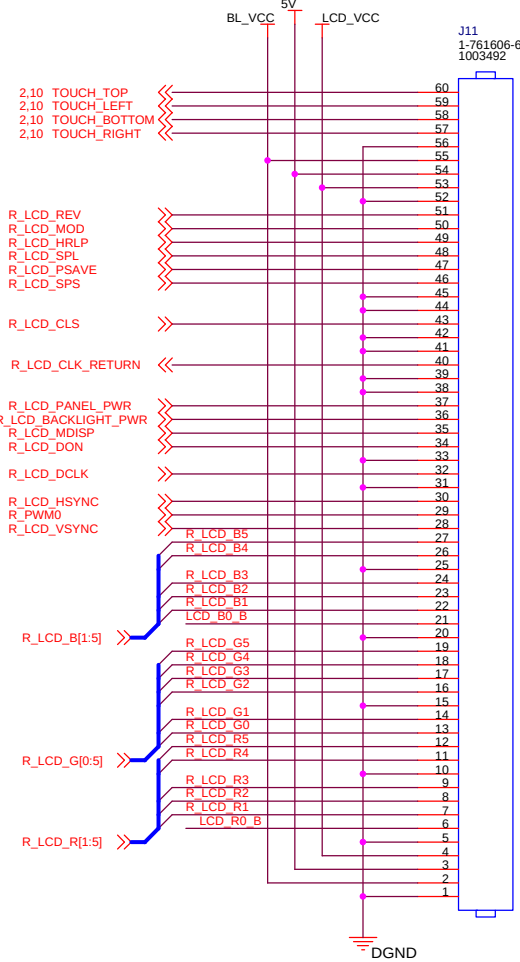
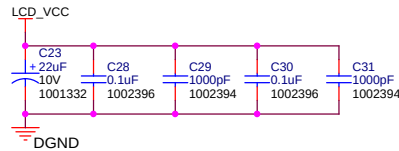
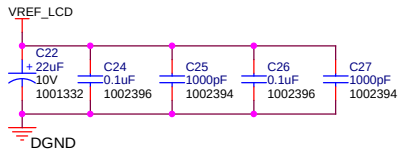
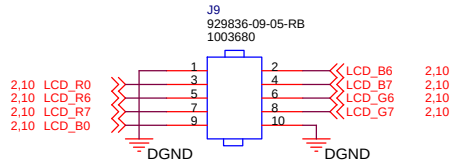
LCD SIGNAL OPERATING VOLTAGE SELECTOR



CUSTOM BACK LIGHT VOLTAGE CONNECTOR



DESIGN NOTE: DEFAULT IS
POPULATED FOR 18BPP PANELS
TO BE DRIVEN ONLY BY 16BPP



DESIGN NOTE: THIS CONNECTOR IS ACTUALLY A 30X2
DUAL ROW .100" PITCH, .025"SQ POST MALE
HEADER. IT IS REPRESENTED SYMBOLICALLY AS A
SINGLE ROW CONNECTOR TO
ILLUSTRATE HOW THE SIGNALS WILL ROUTE THROUGH
THE RIBBON CABLE.

07 - LCD

08 - POWER

DESIGN NOTE: DEFAULT IS J39 NOT POPULATED

DESIGN NOTE: DEFAULT IS J39
NOT POPULATED

2,9,10 MSTR_nRST

2,10 uP_GPIO_2

R56
10K
5%
1001336

Q2
N-CH
2N7002-7-F
1001604

S1
KSC421GB (RoHS)
1002514

DGND

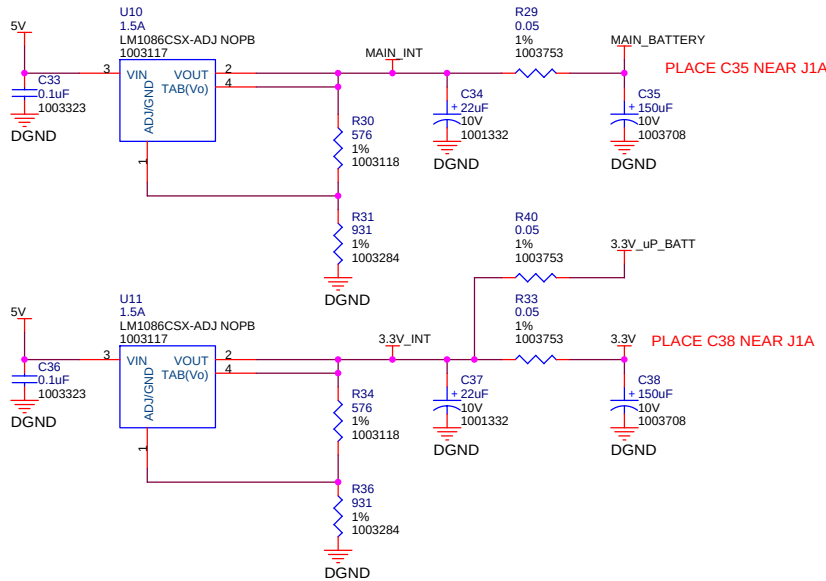
DGND

DGND

DESIGN NOTE: uP_GPIO_2 CAN BE USED TO RESET THE BOARD USING SW

DESIGN NOTE: uP_GPIO_2 CAN BE USED TO RESET THE BOARD USING SW

The diagram shows three LEDs connected to the 3.3V pin through 330 Ohm resistors (R32, R35, R43) and green LEDs (D3, D4, D6). The LEDs are connected to DGND. The control signals are GPIO 0, GPIO 1, and POWER ON.

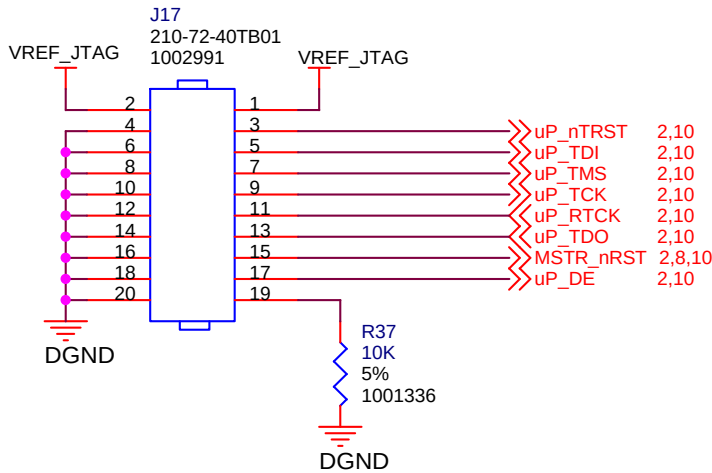


PLACE C35 NEAR J1A

PLACE C38 NEAR J1A

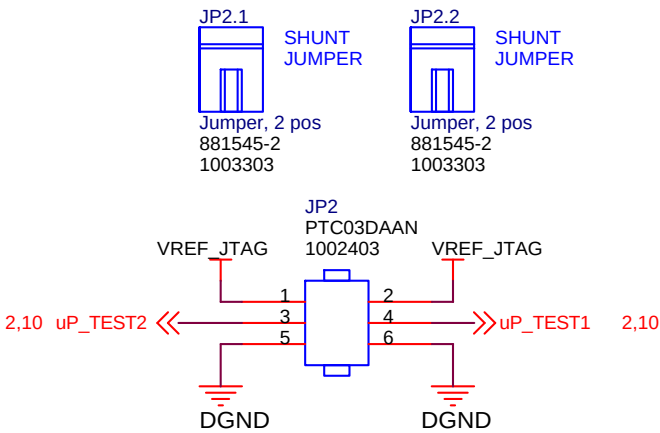
ARM JTAG

DESIGN NOTE: NOT APPLICABLE TO ALL PRODUCTS



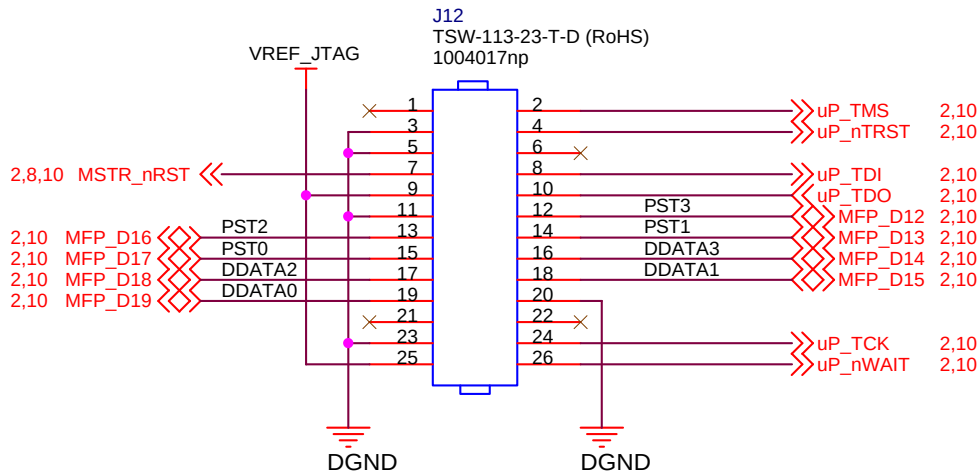
JTAG MODE

DESIGN NOTE: NOT APPLICABLE TO ALL PRODUCTS

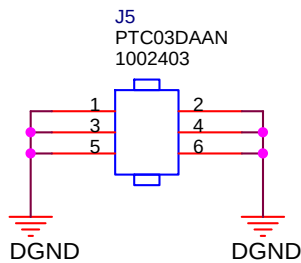


COLDFIRE JTAG

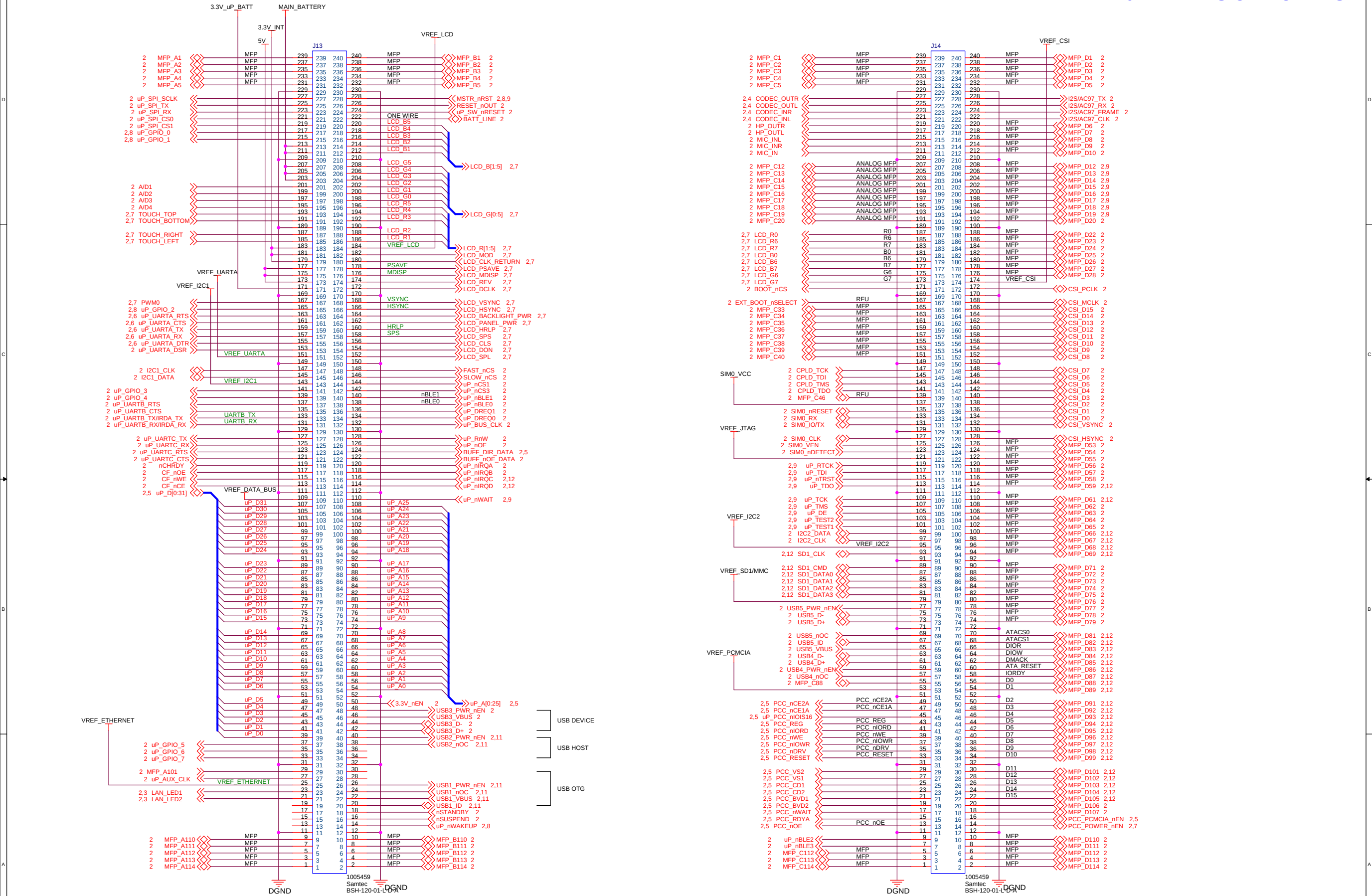
DESIGN NOTE: NOT APPLICABLE TO ALL PRODUCTS



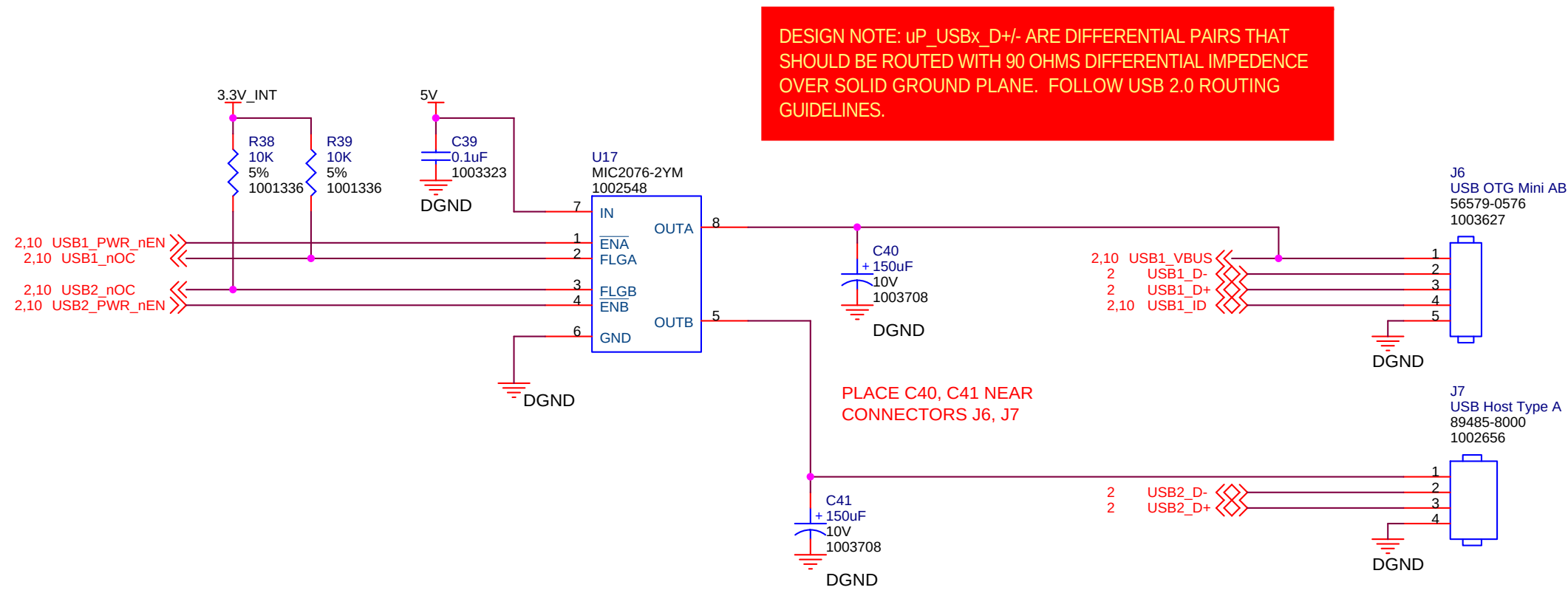
PROBE GND CONNECTOR



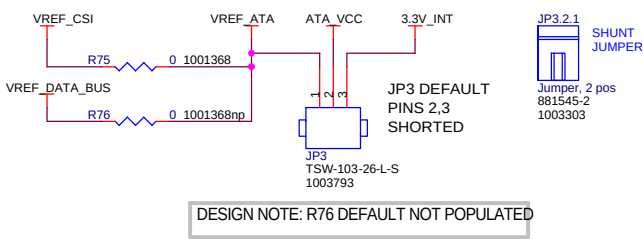
10 - EXPANSION BOARDS



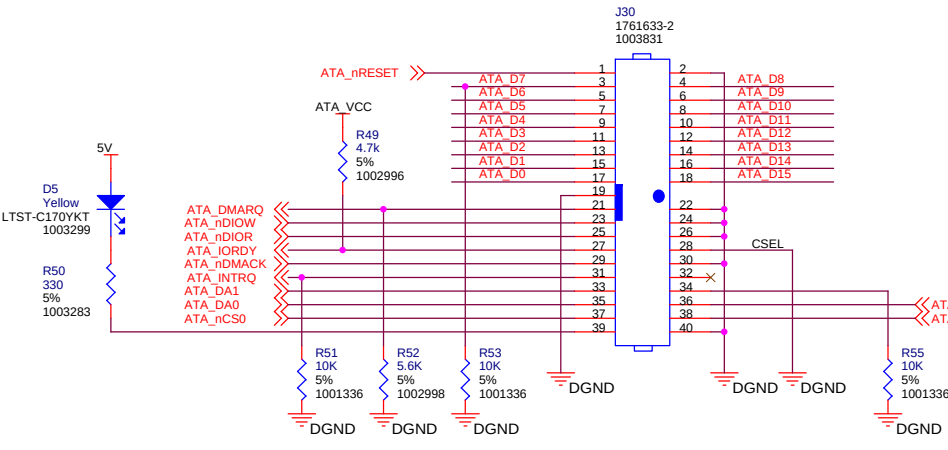
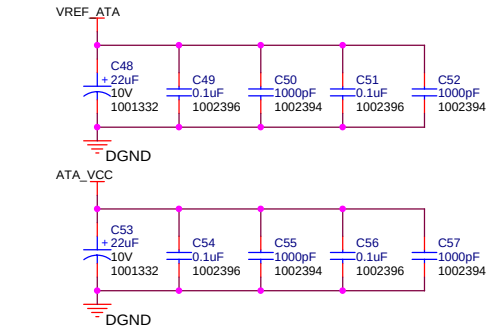
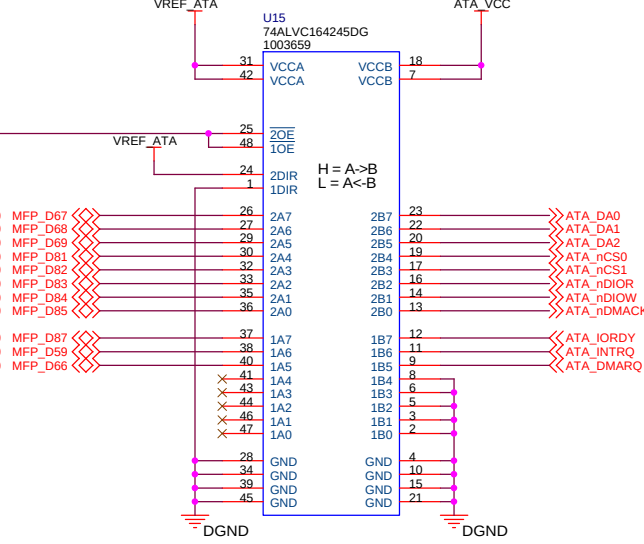
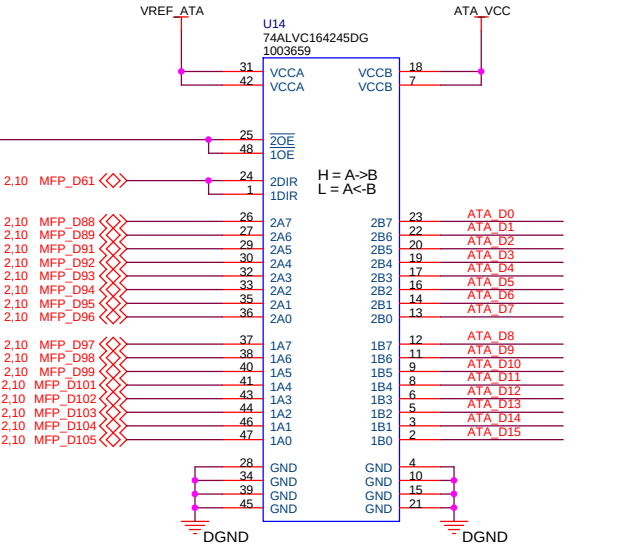
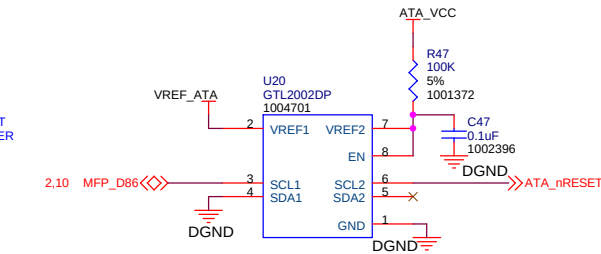
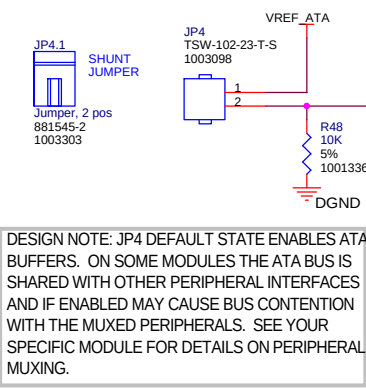
11 - USB



ATA OPERATING VOLTAGE SELECTOR

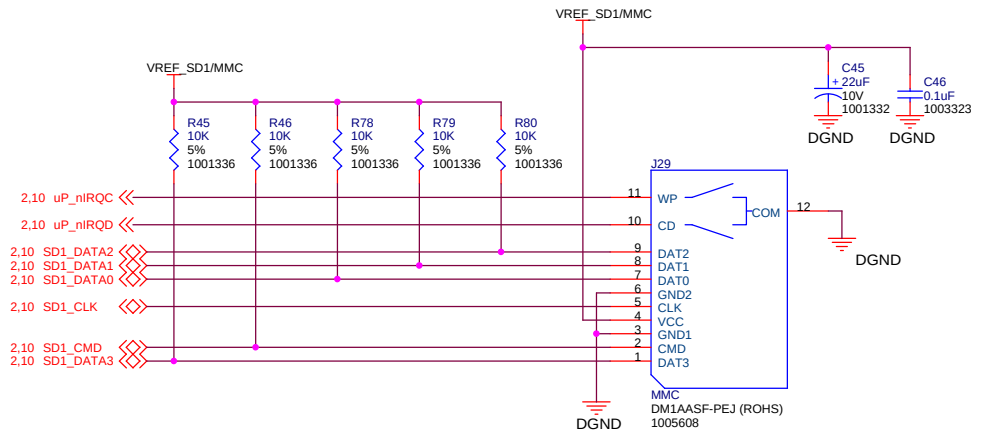


ATA SIGNAL ENABLE



SD/MMC

DESIGN NOTE: PULLUPS FOR uP_nIRQC, uP_nIRQD ARE FOUND ON MODULE



13 - ECO LIST

Production Changes v5.0 - vA		
Mantis #	Pages	Description
0002858	6	The UART to USB chip backpowers the SOM when the baseboard is off. Following the proven fix on the SDK-COM-Express, this was implemented on this design.
0003417	all	All of the 3.3V circuitry on the SDK2 board uses power from the 3.3V plane, not the 3.3V_INT plane (in fact, the 3.3V_INT net doesn't even connect to anything). The primary issue with this is that it is not possible to measure 3.3V current draw on the SOM by monitoring voltage over R33. The voltage over R33 depicts current draw of the SOM as well as current draw of the 3.3V components on the baseboard. The whole idea of the current measurement resistors is to isolate the power for the SOM so power measurements can be taken. If I'm not missing something here, I'd recommend that we fix this in later revisions of the SDK2 board. The functionality/systems impact is nil - there are no differences as far as function/operation with the exception of the ability to measure current consumption for the SOM only. Updated SDK2 by changing the power distribution structure such that 3.3V_INT is used for all SDK2 and Expansion Connector power and that 3.3V is used only for the SOM.
0003404	12	Pull up resistor on CMD line of MMC is not strong enough. Change R46 to 10K. Add 10K pull up resistors to MMC D0 through D3.
0003457	8	Changed out part U23 for one with a lower trip voltage. Made default population to be populated, added 2x 150uF caps on the 5Vin rail to help minimize the droop in the 5V rail when the power button is asserted.
Production Changes vA - vB		
0004239	3, 8	Added note for LAN9211 IC. Added note to power page to reference design guide.
0004130	3	Fixed deisgn note to call out 100 Ohm impedance.