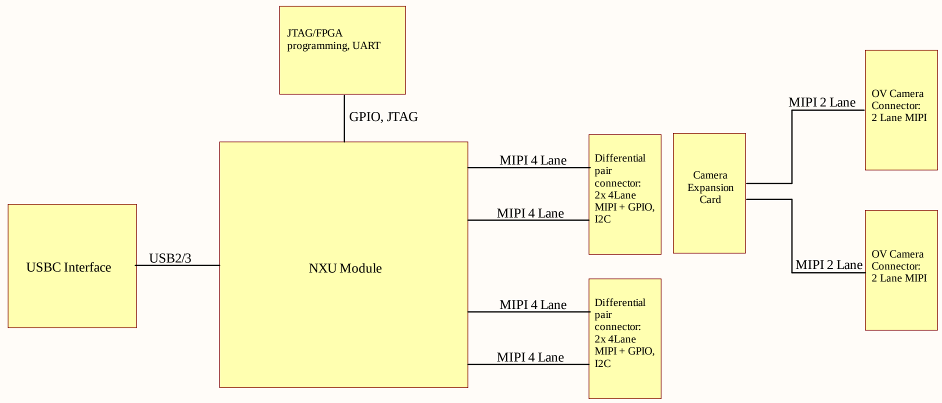
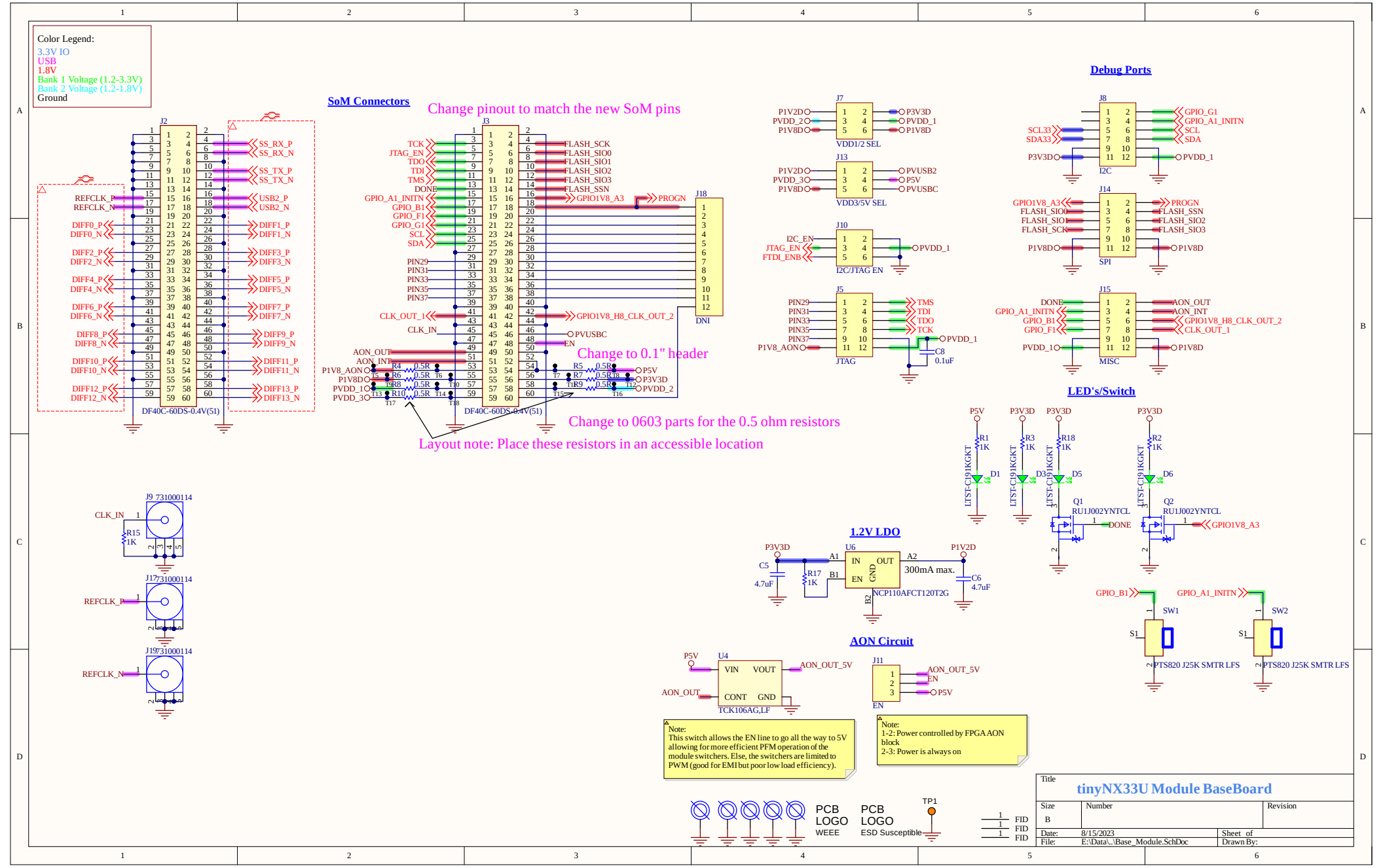
REVISION	DESCRIPTION	DATE	APPROVED



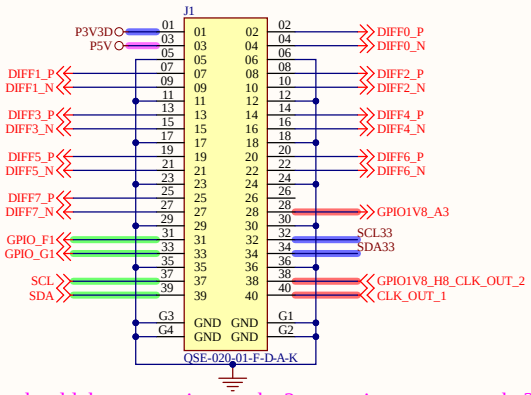
APPROVALS	DATE
ENG: *	
DSN: *	
CHK: *	

REFERENCE DOCUMENTS
BOM:
ASSY DWG:
FAB DWG:
PCB DWG:

PROJECT: Altium
PROJECT REVISION: DOCUMENT REVISION: DESIGN ITEM:
TITLE: *
SIZE: B | CAGE CODE | DWG NO. | REV
SCALE: | FILE NAME: Overview.SchDoc | SHEET 1 OF 4


Color Legend:
3.3V IO
USB
1.8V
Bank 1 Voltage (1.2-3.3V)
Bank 2 Voltage (1.2-1.8V)
Ground
SoM Connectors
Change pinout to match the new SoM pins
Change to 0.1" header
Change to 0603 parts for the 0.5 ohm resistors
Layout note: Place these resistors in an accessible location
Debug Ports
LED's/Switch
1.2V LDO
AON Circuit
DF40C-60DS-0.4V(51)
TCK106AG,LF
NCP110AFCT120T2G
RU1J002YNTCL
LTST-C191KGKT
PTS820 J25K SMTR LFS
tinyNX33U Module BaseBoard
PCB LOGO
WEEE
ESD Susceptible
TP1

Expansion Connector A



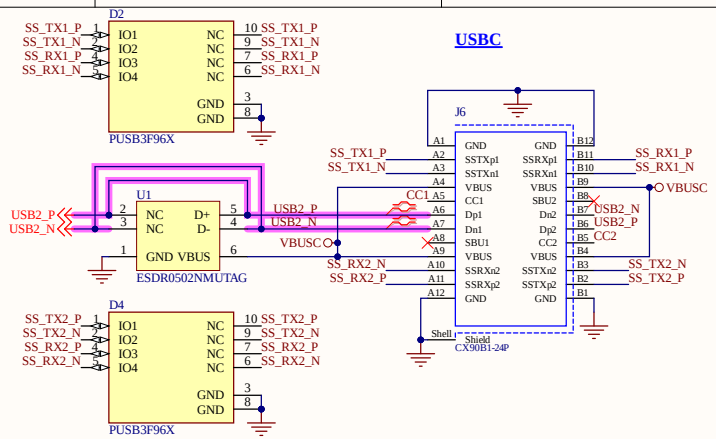
MIPI Possibilities with 6 Lanes:
- 1x 4 Lane
- 2x 2 Lane
- 3x 1 Lane

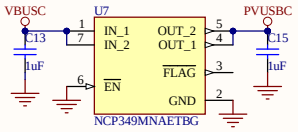
MIPI Possibilities with 14 Lanes:
- 2x 4 Lane + 1x 2 Lane
- 4x 2 Lane + 1 Lane
- 7x 1 Lane

and many others

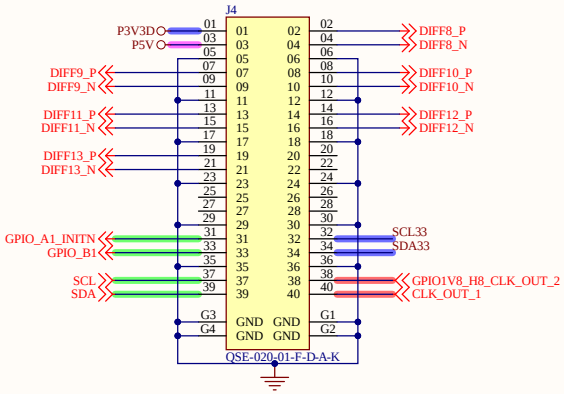
What should the connections to the 2 expansion connectors be?
Should we add the unconnected pins from the SoM here for future use?

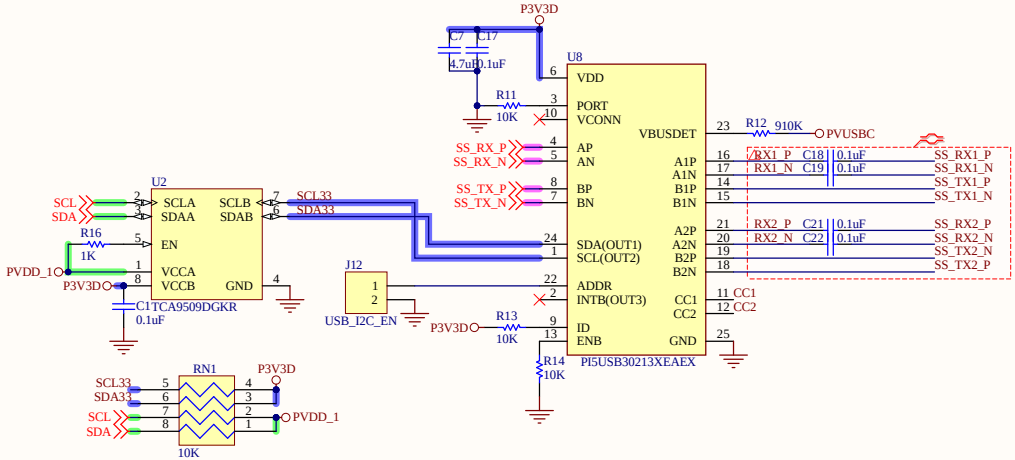
USBC



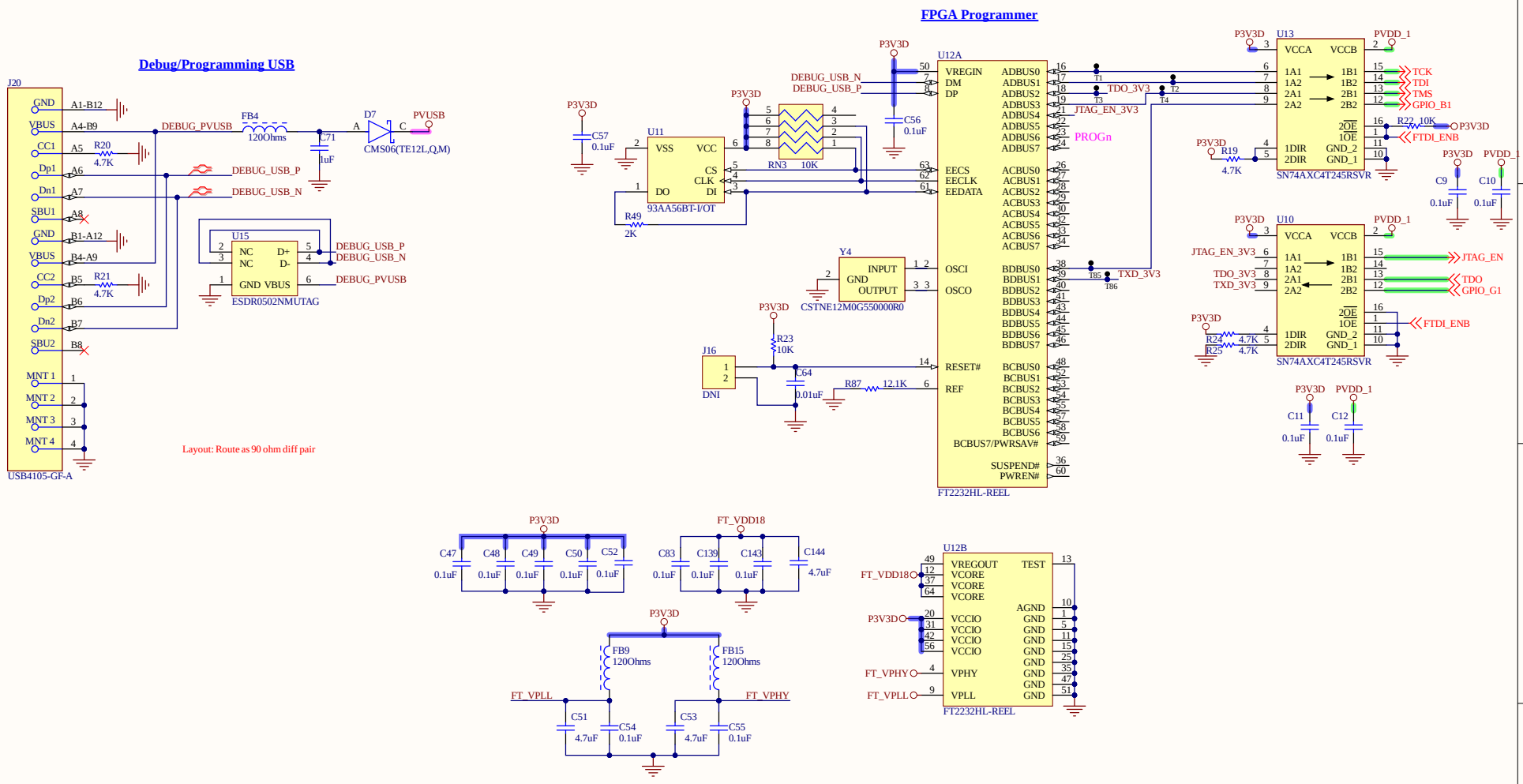


Expansion Connector B





Title	tinyNX33U Module BaseBoard	
Size: B	Number:	Revision:
Date: 8/15/2023		Sheet of
File: E:\Data\..\Base_USB_Camera.SchDoc		Drawn By:



Title	FPGA Programmer		
Size: B	Number:	Revision:	
Date: 8/15/2023		Sheet of	
File: E:\Data\..\FPGA_programmer.SchDoc		Drawn By:	

Board Stack Report