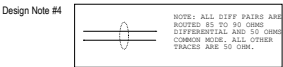


TUSB422EVM INT057 Rev. A

Design Note #1 Keep the USB2 DPIDM stub as short as possible(less than 2.5mm recommended)
Design Note #2 Place LP1, LP2, LP3, at least 5mm away from U1 socket outline
Design Note #3 C12, C13 Test Purposes Only

Design Note #24 PLACE THESE ANY WHERE ON THE BOARD

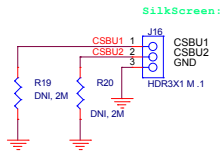
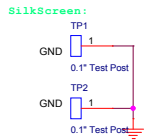


Design Note #5 Place near U1
Design Note #6 Max Value 220uF
Design Note #7 Place Vconn LED on/towards the edge
Design Note #8 Recommended for ESD protection.
Design Note #9 Place on the edge
Design Note #10 Place near J3
Design Note #11 If an external DC controller to be used through J1, the 5V_VBUS_EN or HV_EN need to be configured through the DIP_SW option. End user must ensure only one option is enabled at a time.
Design Note #12 Dual Footprint with Socket
Design Note #13 See the layout guideline for the 20pin header placement dimension/guidelines.
Design Note #14 Place on the edge of the board. Straight or RT angle
Design Note #15 NOTE: POPULATE JUMPER BY DEFAULT
Design Note #16 Programmable up to 20V VOUT = 15V@5A
Design Note #17 SRC/SNK/DRP Mode - SWITCH must be in 1-2 position with DC 20V supply
SNK Mode - Switch can be in position 1-2 with 20V DC IN or position 2-3 with VBUS supplied from SRC.
Design Note #18 Place LEDs on/towards the edge of the board.
Design Note #19 VIN range = 4.5V-22V
Design Note #20 VIN = 2.5-5.5V
Design Note #21 Vout = 4.25V
R28 = 1.8M
R29 = 200K
Design Note #22 Vout = 5V
R28 = 1.8M
R29 = 200K
Design Note #23 VIN range = 6V-20V

 **TEXAS INSTRUMENTS**

TUSB422EVM INT057 - Rev. A

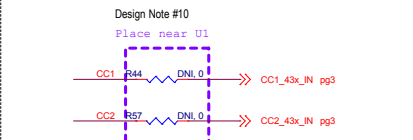
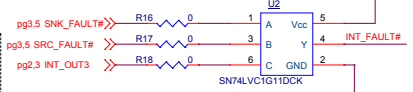
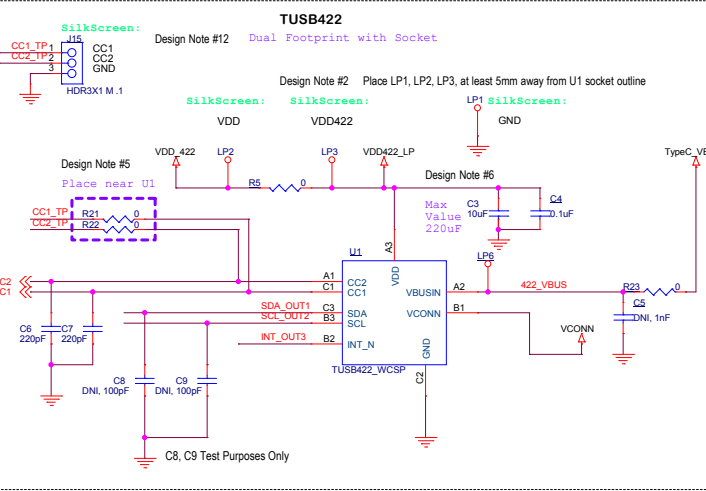
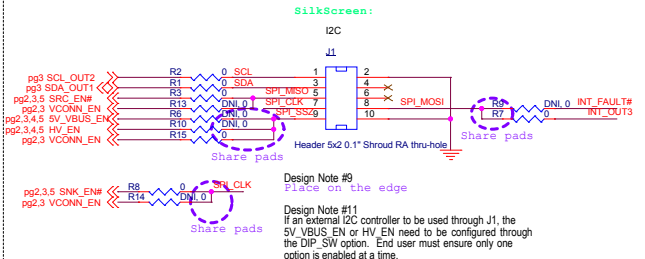
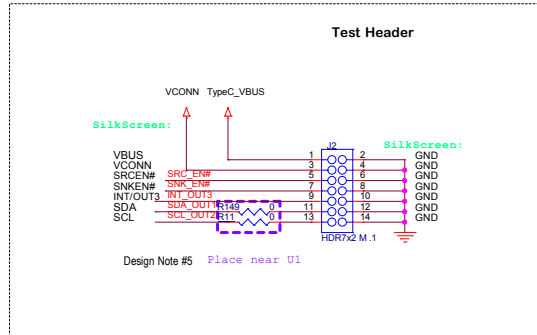
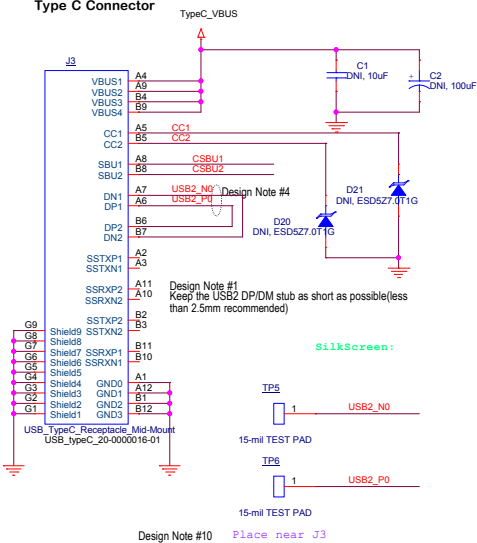
SIZE Custom	DWG NO.
SCALE: NONE	Friday, August 05, 2016
Sheet 1	of 5



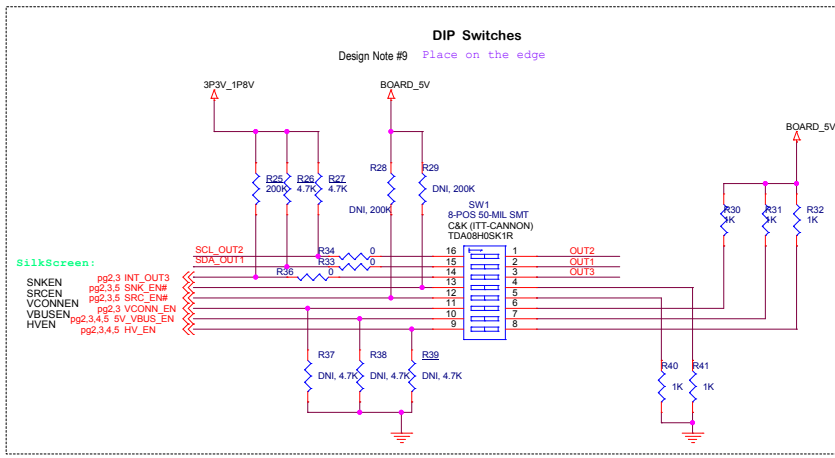
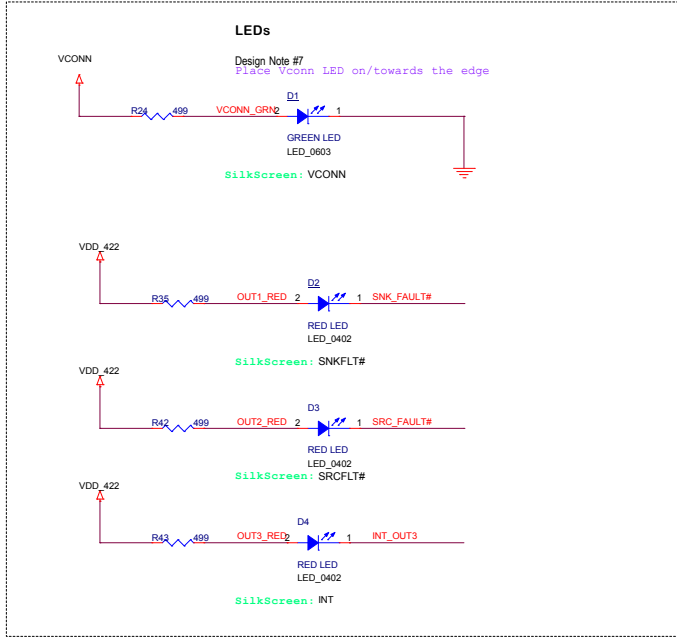
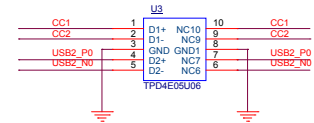
Type C Connector

TypeC Connector Pin Mapping

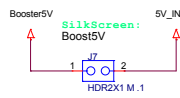
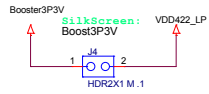
GND	A1	B12	GND
SSTXP1	A2	B11	SSRXN1
SSTXP2	A3	B10	SSRXN2
VBUS	A4	B9	VBUS
CC1	A5	B8	SBU2
DP1	A6	B7	DN2
DN1	A7	B6	DP2
SBU1	A8	B5	CC2
VBUS	A9	B4	VBUS
SSRXN2	A10	B3	SSTXN2
SSRXN1	A11	B2	SSTXN1
SSRXN2	A12	B1	GND
SSRXN1	A13	B1	GND



Design Note #8
Recommended for ESD protection.

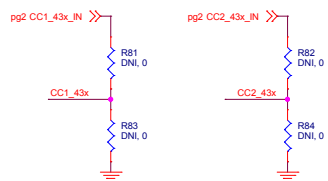
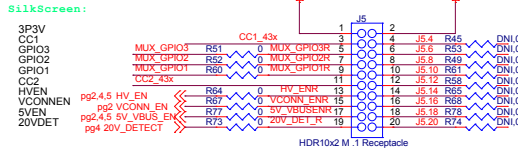
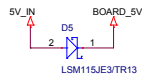


Boosterpack Header

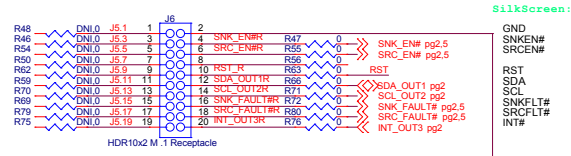


Design Note #14 Place on the edge of the board.
Straight or RT angle

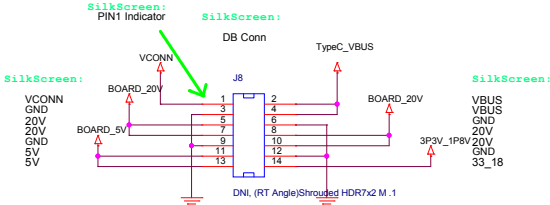
MUST BE OPEN unless FB1 removed



Design Note #13 See the layout guideline for the 20pin header placement dimension/guidelines.



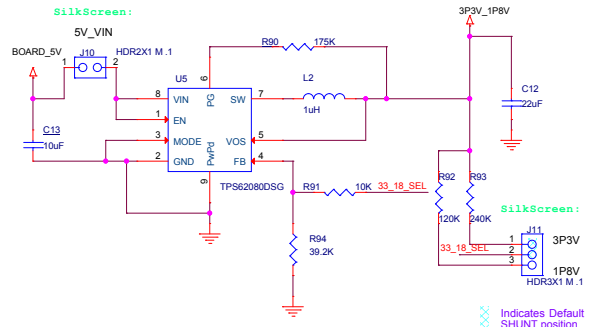
DaughterPWR Header



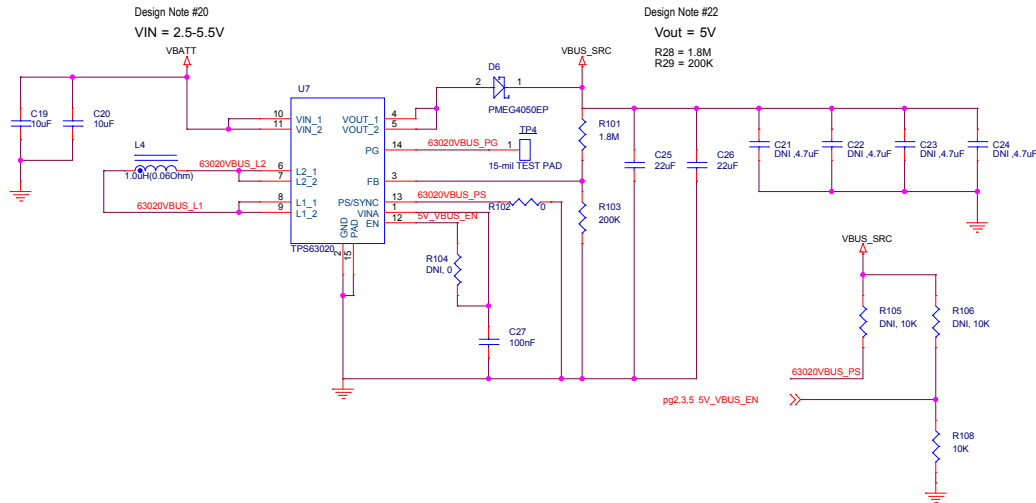
Design Note #9 Place on the edge

3P3V/1P8V Buck

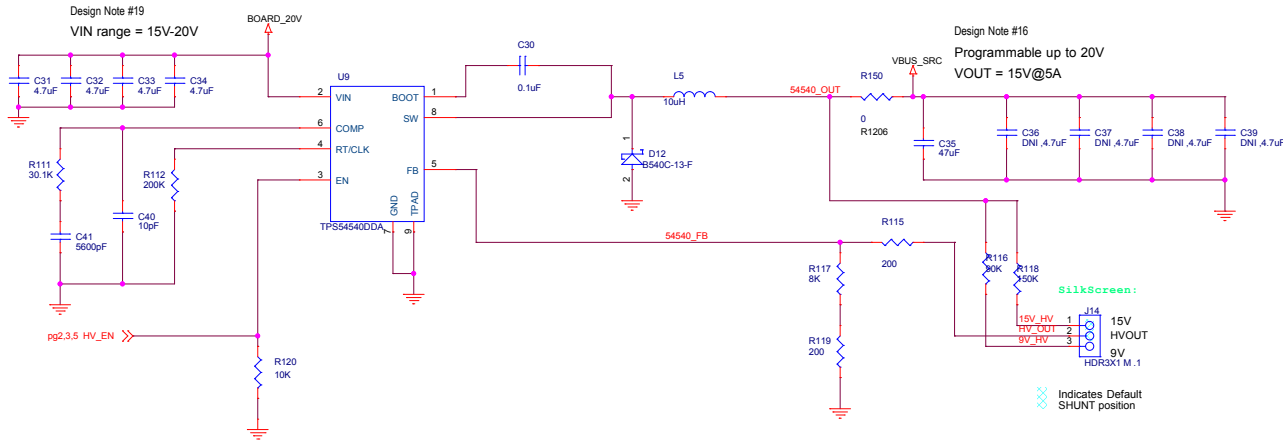
Design Note #15 NOTE: POPULATE JUMPER BY DEFAULT



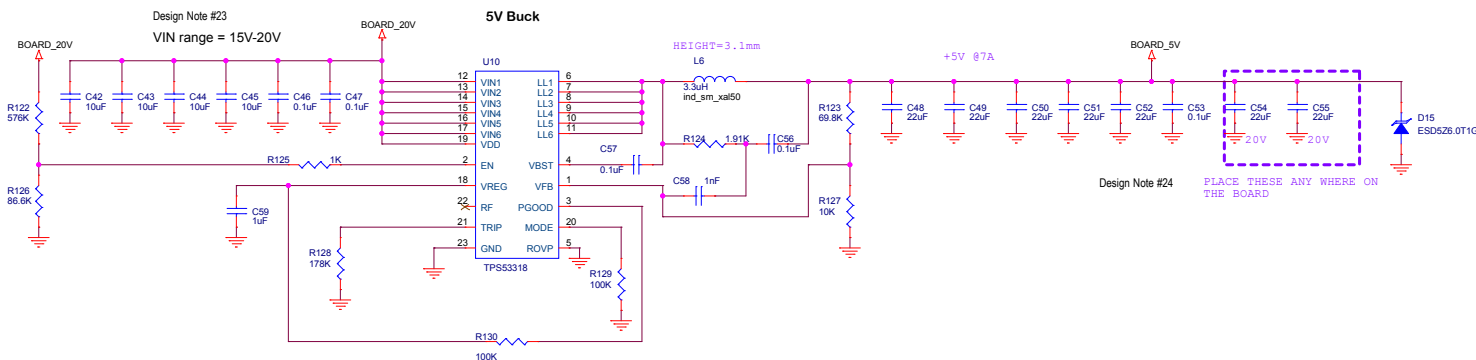
5V VBUS_Buck/Boost



15V Buck

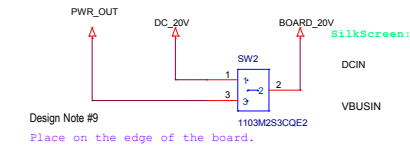


5V Buck

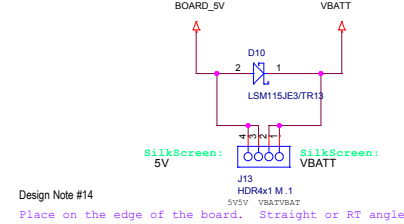


BOARD POWER IN SWITCH

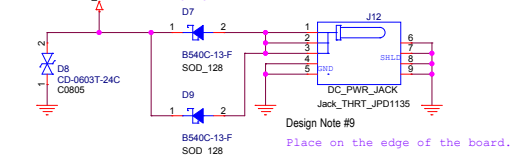
Design Note #17
SRC/SNK/DRP Mode - SWITCH must be in 1-2 position with DC 20V supply
SNK Mode - Switch can be in position 1-2 with 20V DC IN or position 2-3 with VBUS supplied from SRC.



Vbat



20V DC IN



POWER LEDs

