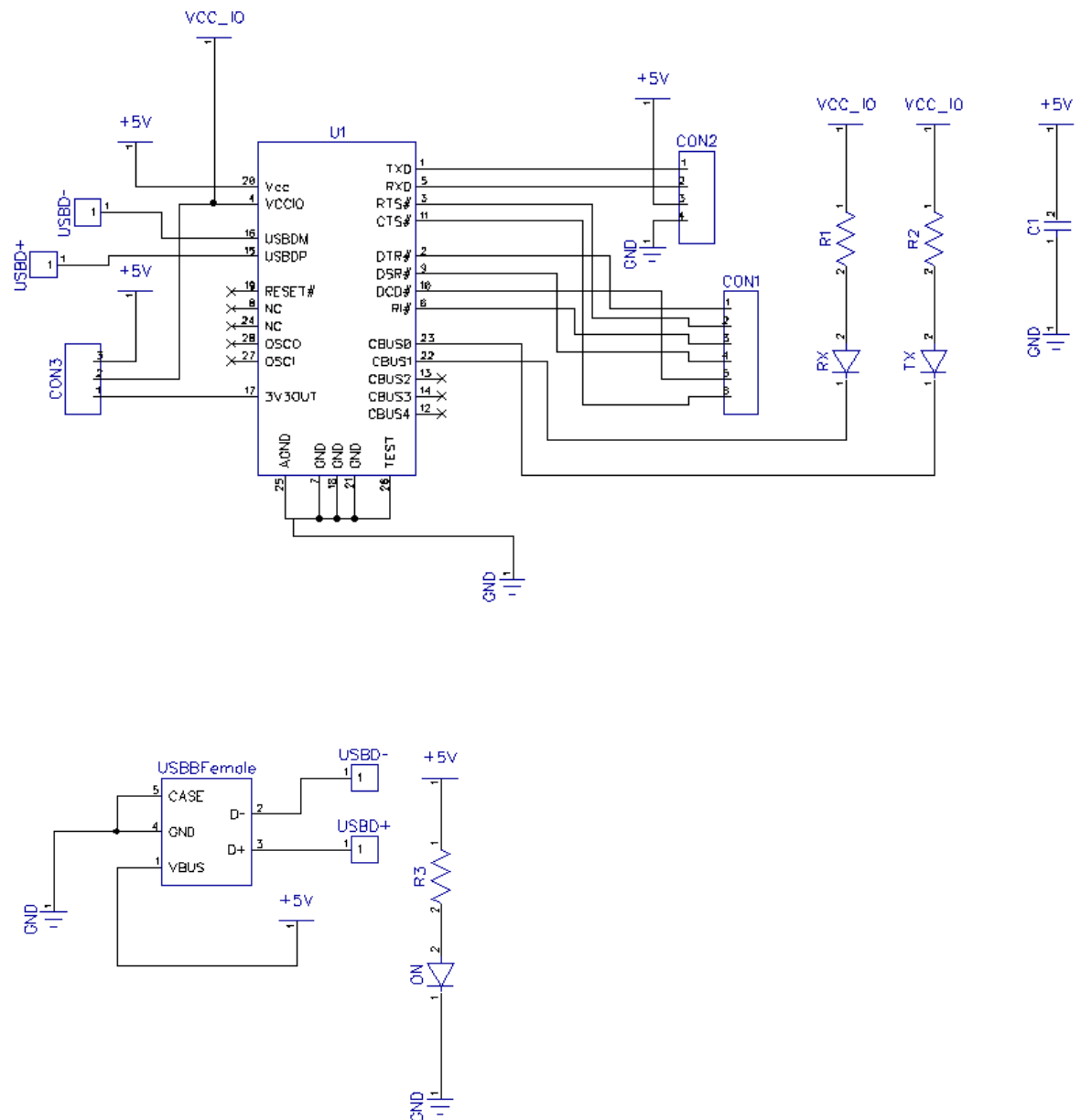# DESD - PCB Designing
DAY 3

1. Recap Day 2 & clear doubts if any from day 2
2. **Assignment : Design double side PCB for FT232 USB to TTL Converter**
   a. Refer Schematic and BoM
   b. PCB : Double Size 60x60 mm
   c. Mounting Holes at four corner : Hole SIze 3.2mm, Ring Size 5mm, Location 3mm inside from edges
   d. Logically place the connectors
   e. Print Connector description on PCB
   f. Creation of panels & gerber files
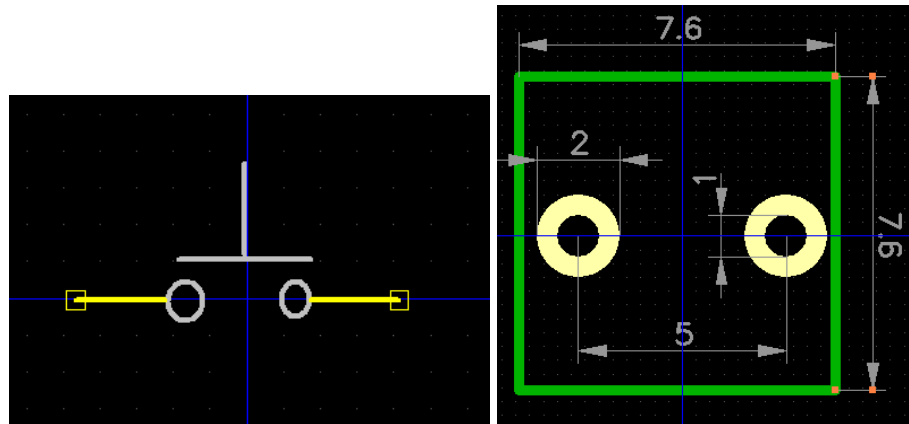
BoM :

RefDes	Value	Name
C1	0.1uF 0805	CAP_0805
CON1	SIP6	SIP-6Pin
CON2	SIP4	SIP-4Pin
CON3	SIP3	SIP-3Pin
ON	RED LED 0805	DIODE_0805
R1	520E	RES_0805
R2	520E	RES_0805
R3	1K	RES_0805
RX	GREEN LED 0805	DIODE_0805
TX	Yellow LED 0805	DIODE_0805
U1	FT232RL	FT232RL
USBBFemale	USB B Type RA Female	292317-4

=

3. Explain Library Creation process
4. Assignment : Edit 0805 LED Component Symbol. Add Arrows.
5. Assignment : Assign Pattern to above edited component symbol.
6. Assignment : Create new Component & Pattern For Switch as per below details

7. Assignment : Refer datasheet of SMLP34RGB1W3 and design the component
8. Assignment : Refer datasheet of TPS73733DCQ and design the component

Trainer : Pratik Deshpande