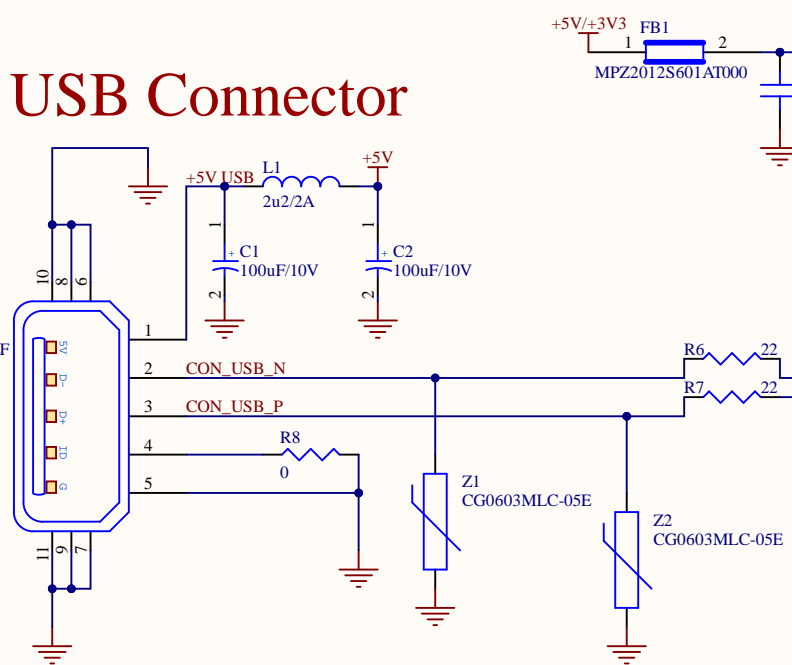
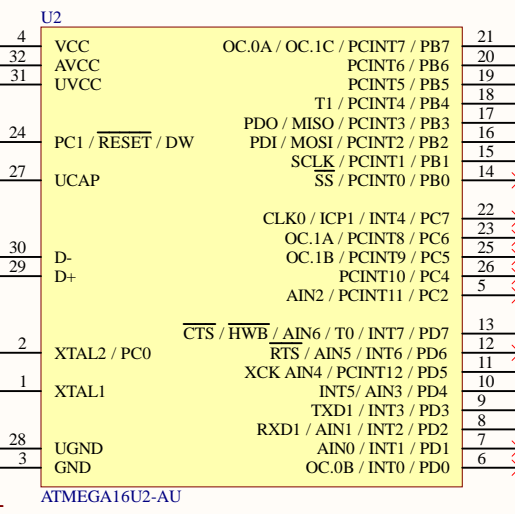


# 28 PINS SCHEMATIC

## Micro USB Connector



## 16U2

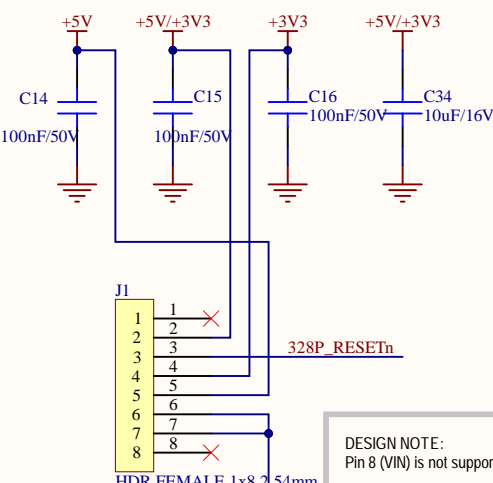


## JP2

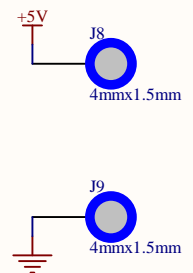
## ICSP1

## JP3

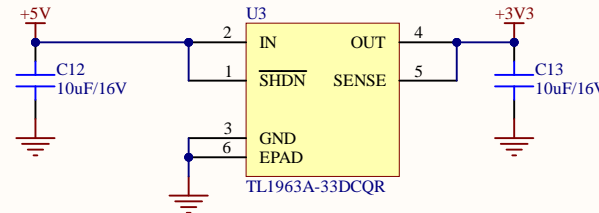
## Power



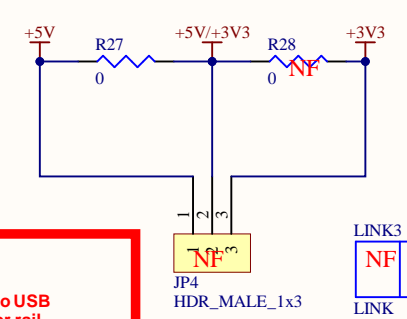
## Power Pads



## 3V3 LDO

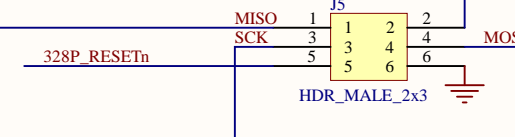


## Power Selection



DESIGN NOTE:  
This board can be powered from micro USB connector (J7) or a single +3.3V power rail (through J1 pin 4). If +3.3V is used, fit R28 and R28. In this case, JP4 & R27 must NOT be fitted, otherwise the board may be damaged.

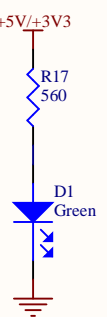
## ICSP2



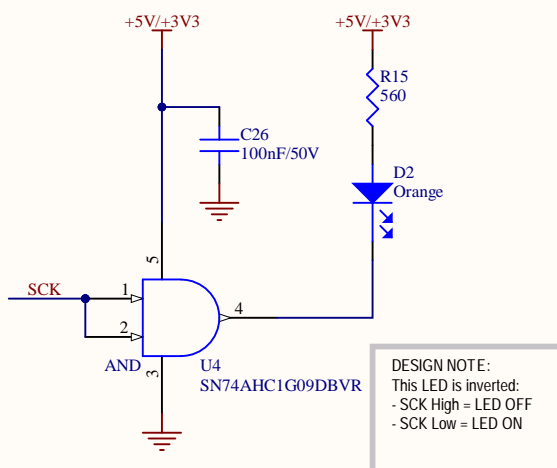
## DIP Socket



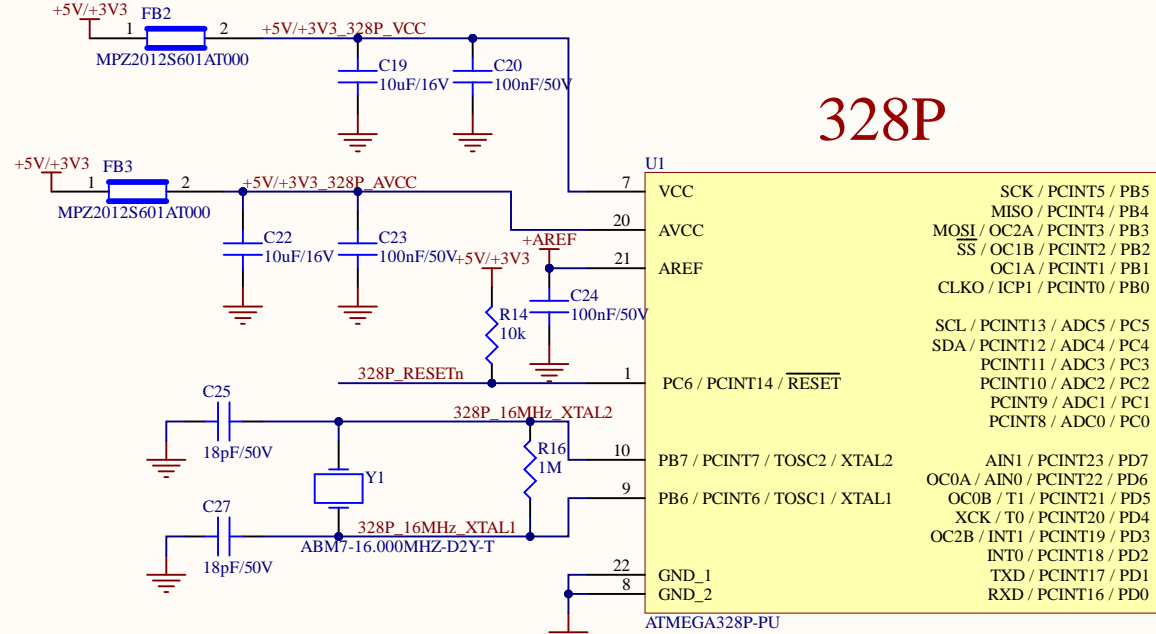
## Power LED



## User LED



## 328P

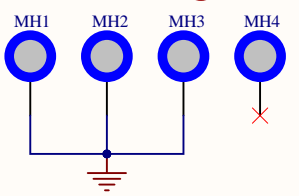


## IOH

## AD

## IOL

## Mounting Holes

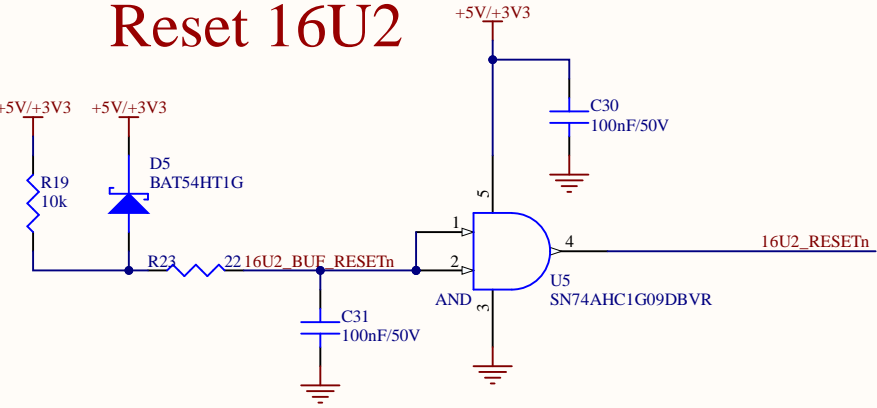


## Fiducials

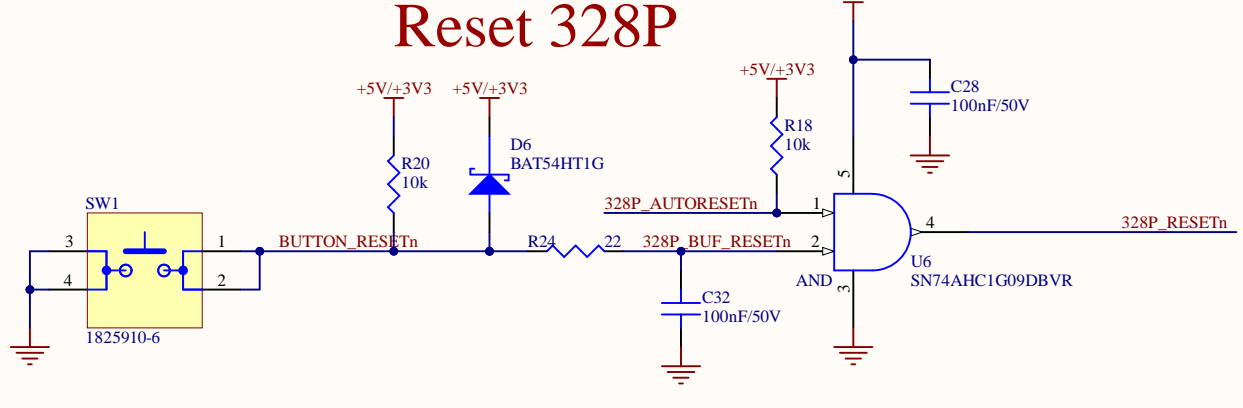


LAYOUT NOTE:  
1) Route all the POWER tracks with minimum track width 0.4mm.  
2) Route all the other tracks by 0.4mm and change them by the end of the design to 0.2mm. To change all of them at once, use this filter: "(not InNet(\*)) and not InNet(GND))" and IsTrack and (OnLayer("L1") or OnLayer("L2"))" and then set 0.2mm width in PCB Inspector panel.

## Reset 16U2



## Reset 328P



## User Button

