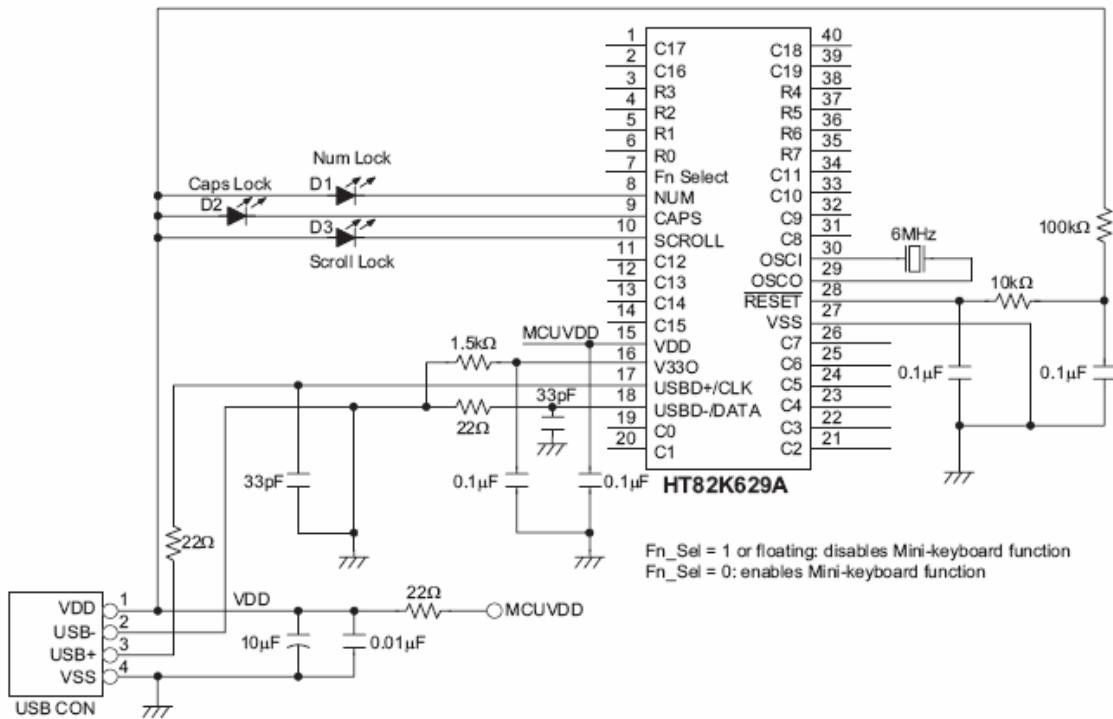


Application Circuits



Note: For single side PCB, the GND should be routed first. Avoid routing the GND line and VDD line with jumping wires.

The GND and VDD lines should be as wide as possible, also it is recommended that the GND and VDD lines are placed in an empty area, in order to increase their area. Wherever possible the GND plane should surround pins such as OSCI, OSCO, VDD, V330, RESET etc and other related circuits to minimize the noise effects.

The GND/VDD loop area should be minimized. Try to keep GND and VDD lines parallel.