How to edit the HelloFTCI file to include an LED an BUTTON

Notes by Ella Peinovich

Save from web: helloFTCI.brd, sch, and lbr Eagle files, also ng.lbr

New > Project > name “helloFTCI” (be sure it is not called “helloFTCI (1)”)

See project path at bottom of screen > save files in project folder

Activate fab.lbr > click green dot (packages of components and parts libraries, right click folder to view components)

Click eyeball to click on schematic and find highlighted on artwork

-Attiny- microcontroller

-Resonator - Time keeper

-AVRISP - Programming header

-JP1 –FTDI header

-Pull-up resistor (for time)

-Decoupling Capacitor (bucket that smoothes out power flow, microcontrollers are very sensitive to power) VCC- power rail, GND –ground rail

Pull-up Resistor-

Running mode (high) or programming mode (low)

* Put resistor on RST – reset line (when Reset is T (true) then it is grounded, assume program does it)

OHM’s Law V=IR

V(volt) = I(amps, amount of current)R(ohms Resistor)

“Node” or “Net” –components assuming no wires connecting them

* Turn on grid “grid on” >Check to see Schematic is on grid > if not, apply “run snap-on-grid-sch” script to fit to grid > check for errors “ERC”
* Check “rats” for ratsnest to see yellow error lines >then reconnect wires

Add LED and Button

-add current loading resistor to diode on path between VCC and ground (only one current path, from VCC to GND, so resistor can be put anywhere)

-hook up to Microcontroller pin, so it can be set to low (sink current from other source) or high (sourcing current from micro controller).

>”add” >find ng.lib >LED > (length and width measurements packages) >add 1206 LED > OK >(right click to rotate) >add 1206 resistor >copy second resistor > add 6mm button

>add 1 switch to look at state, and 1 to power/drive LED (use open/ unused pins)

>place resistor on LED >type “net” >connect pin of LED to resistor >add air wire on other side of resistor >copy any GND to other end of airwire

>stretch back PB1 > add wire to PB2 and LED >click on wires and name both “LED” >click ‘yes’ to connect wires >use label (abc) button to add label name to wire

>”group” linked components >make box around group >right click to “move group”

>add pullup resistor >copy VCC and GND >add “button” name/connection to PA3 and end of wire from 1 pin on switch

>fix board >arrange LED and its Resistor first (right click to rotate), type “rats” to simplify wires

>type “wire” to size wire to .01 for smallest wires, can also choose angles >use “route” button to select airwire for connection, use “ripup” to remove routed wire connection

>use group to select right side of board >right click to “move group” >pull orthogonally

>load drc file > type ”drc” to check for errors >load drc preset >apply >check >look for error message at bottom left of screen

>”disp” , select ‘Top’ only > ok> export > image >select top only >save out name “ > type in 600 dpi, monochrome

>same process for ‘Dimension’ only >once saved, delete circles in paint program

(NOTE: if can see all, but cannot move anything, be sure ‘tOrigins’ layer is on)