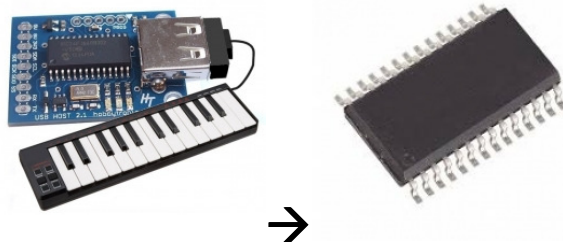


Synth Project References for PCB Layout

USB Host Controller



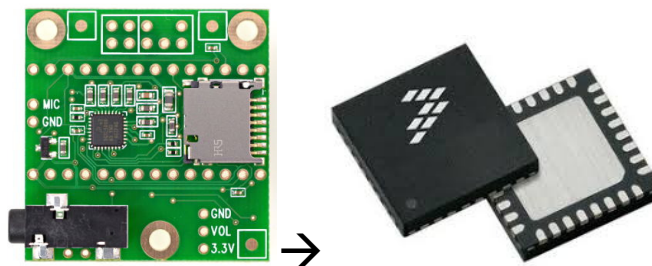
IC - SMD (SOIC) 28 pin:

<http://www.hobbytronics.co.uk/usb-host/usb-host-soic>

Minimal schematic:

<http://www.hobbytronics.co.uk/datasheets/usb-host-v2-basic-IC.pdf>

Teensy Audio Board



Schematic (version 1):

https://www.pjrc.com/store/teensy3_audio.html

SGTL5000 datasheet:

<https://www.pjrc.com/teensy/SGTL5000.pdf>

SGTL5000XNAA3 Symbol & Footprint from NXP:

<https://www.nxp.com/part/SGTL5000XNAA3>

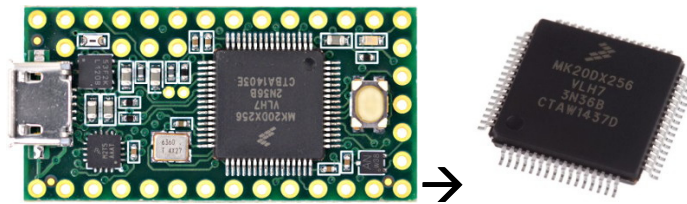
KiCad open-source Library:

<https://github.com/marshalltaylorSFE/KiCad-Library/blob/master/SparkFun-IC-Analog.lib>

KiCad open-source Footprint:

https://github.com/marshalltaylorSFE/KiCad-Library/blob/master/SparkFun-Footprints.pretty/SF-QFN-32-NXP-SGTL5000.kicad_mod

Teensy 3.2 uController



Teensy 3.2 Pinout Reference:

https://www.pjrc.com/teensy/card7a_rev1.pdf

Teensy Schematic:

<https://www.pjrc.com/teensy/schematic.html>

KiCad Teensy Library:

https://github.com/XenGi/teensy_library

KiCad Teensy Footprint:

<https://github.com/XenGi/teensy.pretty>

mk20dx256vlh7 Footprint & Symbol (from NXP):

<https://www.nxp.com/part/MK20DX256VLH7>

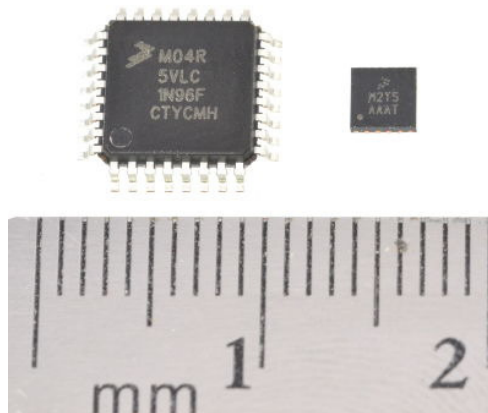
mk20dx256vlh7 Footprint & Symbol:

<https://componentsearchengine.com/mk20dx256vlh7/Nexperia>

Teensy 3.2 DIY Reference Board:

https://www.oshpark.com/shared_projects/d3J03Zeb

Bootloader Chip for DIY Teensy Board MKL04/02



Pre-programmed Chip:

https://www.pjrc.com/store/ic_mkl02.html

KiCad Library:

<https://forum.pjrc.com/threads/30636-MKL02-MKL04-Bootloader-Chips-Available>

MKL04Z32 Symbols & Footprint:

<https://www.snapeda.com/parts/MKL04Z32VLF4/Freescale%20Semiconductor/view-part/?ref=me&t=MKL04>



Free PCB Footprints and Schematic Symbols
(Datasheets, Symbol, Footprint and 3D model)

<https://www.snapeda.com/home>

Import Guide

<https://www.snapeda.com/about/import/?ref=e>

Interesting Tutorials on drawing PCB Layout

- **KiCad User Guide:**
<http://docs.kicad-pcb.org/stable/it/pcbnew.pdf>
- **KiCad Interactive Router:**
<https://www.youtube.com/watch?v=CCG4daPvuVI&feature=youtu.be>
- **Introduction and KiCad Project Creation:**
<https://www.youtube.com/watch?v=iTyi3RvNoB0&feature=youtu.be>
- **Creating Power and Ground Planes in KiCad:**
<https://www.youtube.com/watch?v=WcdJ7FAmD7k&feature=youtu.be>

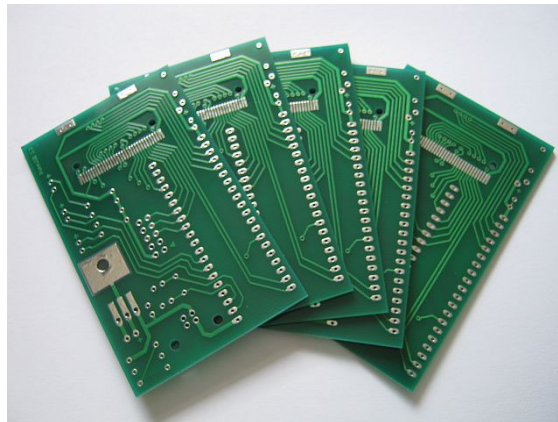
[tu.be](#)

- **Create PCB Footprint component:**
<https://www.youtube.com/watch?v=aVNMJVaRf6M&feature=youtu.be>
- **KiCad Quick-Start Tutorial:**
<https://www.youtube.com/watch?v=zK3rDhJqMu0&t=939s>
- **Board Design Guideline:**
<https://www.altronmfg.com/pcb-design-for-manufacturability>
- **USB High Speed PCB Layout:**
<https://it.emcelettronica.com/pcb-tutorial-lineguida-i-pcb-consegnali-usb-high-speed>
- **KiCad Getting to Blinky 4.0:**
<https://www.youtube.com/playlist?list=PL5iUxv3Op2fOpVASHvcpM2O4UO0yMfKJi>
- **Contextual Electronics tutorials:**
<https://www.youtube.com/user/contextualelectronic>
- **Pad Requirements for SMD Footprints:**
<http://kicad-pcb.org/libraries/klc/F6.3/>
- **KiCad Tutorial (HackaDay Part 3):**
<https://hackaday.com/2016/12/23/creating-a-pcb-in-everything->

[kicad-part-3/](#)

- **Maximize your Production yield (From Altium):**
https://www.altium.com/webinars/on-demand/tech-briefing-dfm-maximize-your-pcb-production-yield?utm_source=ownedmediawebinars&utm_medium=webinar&utm_campaign=webinars
- EMI PCB Layout:
<https://learnemc.com/pcb-layout>

PCB Manufacturing



- “OSHpark” PCB Stencil Manufacturing:
<https://www.oshstencils.com/#>