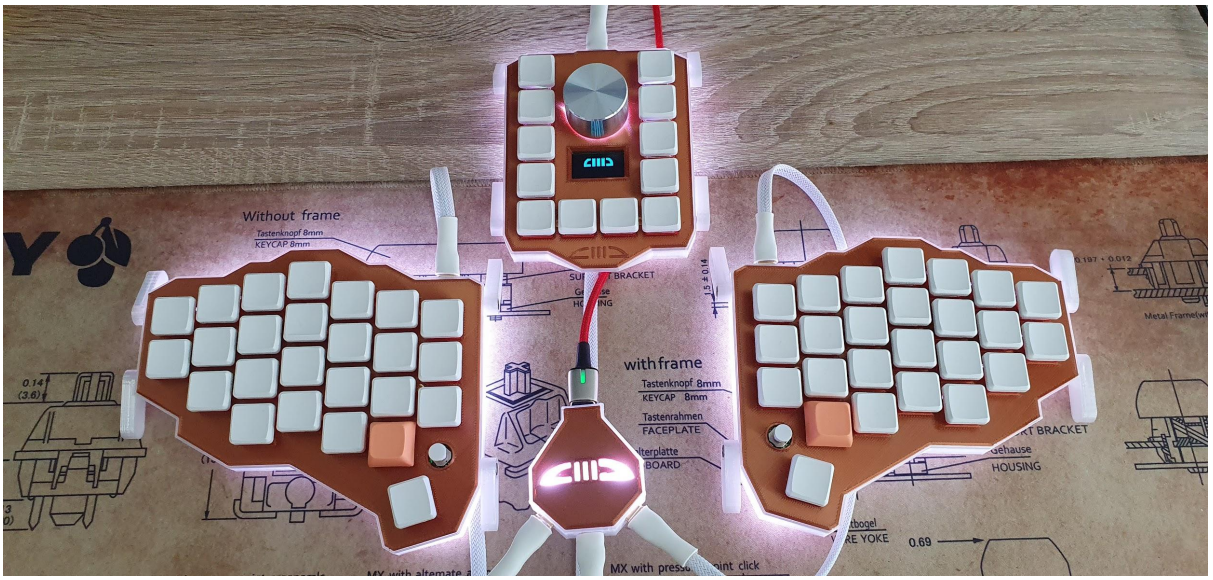
IIICC by kbjunky & Sean Yin

3 piece split keyboard

Assembly Guide



Bill of materials

Item	Size	Count
22 Ohm	R0805	2
330 Ohm	R0805	1
1.5k Ohm	R0805	1
100nF	C0805	51
10k Ohm	R0805	2
10uF	C0805	1
Oscillator 16MHz	OSC-SMD_4P-L3.2-W2.5-BL	1
1k Ohm	R0805	3
1N4148W T4	SOD-123_L2.8-W1.8-LS3.7-RD	75
1uF	C0805	1
22pF	C0805	2
3.3k Ohm	R0805	1
4.7k Ohm	R0805	3
4.7uF	C0805	1
5.1k Ohm	R0805	2
Atmega32U4-AU	QFP-44_L10.0-W10.0-P0.80-LS13.2-BL	1
USB-C		7
MCP23018-E/SS	SSOP-24_L8.1-W5.3-P0.65-LS7.8-BL	3
Encoder	SW-TH_PEC11R-4220F-S0012	1
Reset button	SW-SMD_L4.0-W2.9-LS5.0	1
5W Switch	SW-SMD_TSW-10-1-11B-T15	2
WS2812B-B		44
OLED 128x64		1
Kailh Choc Hot Swap Socket		50/62
Switches Choc V1/V2/X-Switch		50/62
Keycaps		50/62
M2 brass flat head screw	6mm	25
M2 brass flat head screw	3mm	25
M2 brass heat insert	5mm	25
M3 hex socket screw	8mm	12
M3 Thumb screw	8mm	12
Adhesive rubber feet	8x2mm	20
Double sided adhesive tape		
Aluminum Feet		14
Interconnect USB-C Cable	0.5m	3
Magnetic USB-C Hub<->PC Cable		1
Encoder Knob	30x17mm	1

About keycap compatibility

All Choc V1 caps should fit including MBK. Good idea is to put XDA on Space and Enter while using Choc V1. This gives a bit more flexibility while pressing those keys (easier to reach with a thumb).

For Choc V2 I have tested only two types:
YMDK XDA - OK

YMDK ABS OEM(?) profile - for this only buy R1 to R3 and use reversed R1 on row 4 (bottom row). R4 keycaps from this store won't hold, seems like they are too high (more like switch/stem being too short).
In terms of keycaps hitting the plate on Choc V2 I have not noticed any problem with that. Seems like there's still tiny space left.

For Kailh X-Switch there are no other keycaps than the one Kailh is selling on their store. Link.

About LEDs and USB-C

I have made available pads for VCC/GND/D+/D-/LED_DATA on the PCB. A USB-C pinout board can be used but most probably it won't fit the case. Maybe in the future I will add a slightly angled case so this might come handy. As for LEDs a WS2812 strip can be used instead of soldering all the capacitors and LEDs.

PCB Ordering

PCBs were created using EasyEDA and can be directly ordered from there. Use default settings when making your order, that is:

Layers: 2
PCB Qty: 5 (min) or more depending on your needs
Delivery format: Single PCB
PCB Thickness: 1.6mm
PCB Color: Green (default) but you can choose any other color available
Surface Finish: HASL(with lead)
Copper Weight: 1 oz
Gold fingers: No
Confirm production file: No
Flying probe Test: Fully Test
Castellated Holes: No
Remove Order Number: No

Each part has to be ordered separately. Left, Right and Hub are mandatory. Macropad is optional.

Tools

Minimum set of tools/materials required for this build:
Soldering iron.
0.5mm Lead Solder.
No clean flux. Marker or syringe.
Multimeter with diode test.
Tweezers for SMD elements.
Isopropyl alcohol for cleaning PCBs. (Got mine at a pharmacy store)
Magnifying glass might come handy especially if it's on an arm. Or anything to help with soldering USB-C ports and ICs.
Hot glue gun to secure USB-C ports to the PCB
Desoldering wick for cleaning of solder when things go wrong.
Brass sponge for cleaning the tip.

Assembly

You should start with the Hub. First solder in USB-C port that connects Hub to the PC. Check for shorts using pads on the PCB.

D+ <-> D-
GND <-> Vcc
VCC <-> CC1 (5.1k resistor pad)
D- <-> CC2 (5.1k resistor pad)

Next solder in the rest of USB ports one at a time checking for shorts on:

VCC <-> RGB
VCC <-> GND
SDA <-> SCL
SCL <-> RGB

After this is done you can proceed to installing Atmega and all the elements around it. For now skip LED diodes and 100nF capacitors located between them and the edge of the board. Once done connect the board to the PC. If you see a message about an unrecognized device most probably there's a short on VCC and or D+/D-. Double check with a multimeter. If all is good you should be able to flash firmware onto it. Follow QMK guide on flashing firmware.
Finish assembly by installing LEDs and accompanying capacitors. Connect again to check if LEDs are working fine. Be sure to have Macropad disabled for the testing phase as without an OLED screen present driver might freeze the controller.

Next assemble either the left or right keyboard part. Leave the macropad last. Steps below apply to all of the parts.
Just like with the Hub first thing that needs to be installed is the USB-C port. Check for shorts using pads on the PCB.

SCL <-> SDA
RGB <-> SCL
Vcc <-> RGB
Vcc <-> GND

After that install I2C expander (MCP23018) and:

Right:

4.7k address resistor
1k Reset pull-up resistor
100nF Vcc capacitor

Left:

1k Reset pull-up resistor
100nF Vcc capacitor

Macropad:

1.5k and 3.3k address resistors
1k Reset pull-up resistor
100nF Vcc capacitor

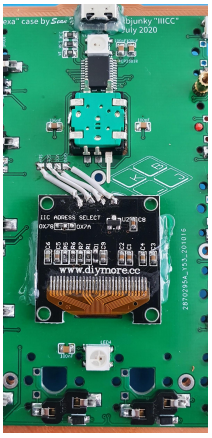
After MCP23018 is installed you should check for shorts on:

Vcc <-> SCL
SCL <-> SDA

At this point it's good to check if communication is working. Install one diode to the nearest switch to the USB-C port. Connect this part to the Hub and use https://keycode.info/ to check if keypress is reported. If not check again for any shorts on the I2C line as well as Vcc/GND. If it works the rest is easy but a bit time consuming. Install all switching diodes, check each key if it works. Then you can proceed to LEDs + capacitors. Best way to install LEDs is in small steps. One or two diodes at a time and check if they work. After that install hot swap sockets. Last but not least, the 5W switch. It doesn't like high temperatures for too long as well as too much solder. Install it as the last element and don't use too much solder/heat. Otherwise it might stop being 'tactile'.

Special note on macropad.

Two things require special steps: an OLED screen and a rotary encoder. OLED should fit inside the cutout. Place it there and use hot glue to glue it to the pcb. Then use a wire to connect OLED to the VCC, GND, SDA, SCL pads nearby.



If your OLED doesn’t work you can try changing its address in firmware. Oddly enough the one I have used while building had the same address as 128x32 which is 3C. 128x64 usually has an address of 0x7C or 0x78 (it’s at the back of the screen).

As for the encoder it’s best to have the top plate printed. Push it through the hole in the plate and tighten with the screw. Now attach the plate to the PCB so that it won't be moving (2 screws will do). Cut off the GND pins (the thick ones on the sides) we won’t be needing them. Position the rotary encoder so that pins align with pads on the pcb (3 bottom ones and 2 on top). Now bend the pins towards the PCB and trim them so that they are touching the PCB. Solder them to the pads and put the washer and the nut that came with the encoder on the top part of the plate to secure it further.

Case assembly

At this point you should have all the pieces working so it's a good time to use hot glue on USB-C ports to secure them to the PCB.

Choose Choc V1/V2 or X-Switch compatible parts. It's possible to print Choc V1/V2 compatible bottom case and use it with an X-Switch plate. Only difference is that the outline of the case will be slightly higher than the plate. It's because the plate for Choc is 2.2mm and for X-Switch 1.6. Also decide if you need support for bottom rubber feet or not (additional M4 holes on the bottom). After printing is done assembly is rather easy. Install heated inserts on the sides. Attach PCB to the top plate with a 6mm flat head screw (from the top). Heated inserts will also be used as spacers on the bottom side of the PCB where the top plate mounting screws are. Once this is done put two pieces together. Top plate should click in. From the bottom screw in 3mm flat head screws to secure top plate and bottom case together.

Hub doesn't need any screws. All parts should fit in tight. Insert PCB into bottom case. First insert USB-C to PC port. Then slide it back a bit so that all the ports align. Then the top plate is two pieces. Transparent bottom one with extruded IIICC logo should fit with the top one in a way that extruded letters go through IIICC cutout in the top part.

Tenting feet

You can choose between 25mm, 20mm and 10mm so it's more of a trial and error before you find the most suitable ones. Use double sided adhesive tape to prevent the keyboard from sliding while tented.

Printing

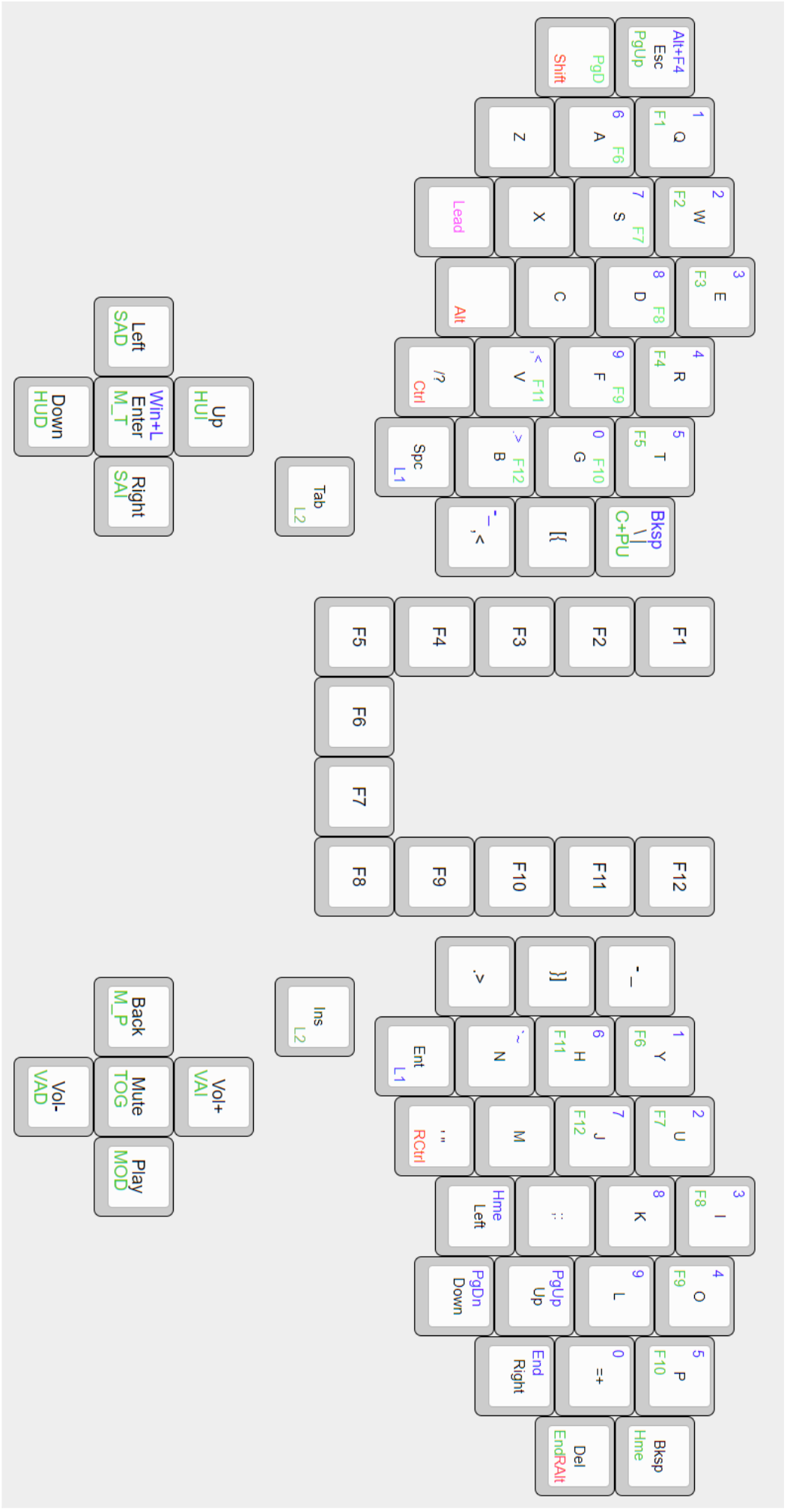
Top plates printed with shiny copper PLA
Layer height 0.2
Infill 100%

Bottom cases printed with translucent PLA
Layer height 0.2
Infill 30%
Support ON (unless your printer can print USB-C cutout without it)

Feet
Layer height 0.2
Infill 30%

5W Switch cap
Layer height 0.2
Infill 100%

Layout


Alt+F4
Esc
PgUp
PgD
Shift
1
F1
Q
6
F6
A
Z
2
F2
W
7
F7
S
X
Lead
3
F3
E
8
F8
D
C
Alt
4
F4
R
9
F9
F
V
/?
Ctrl
5
F5
T
0
F10
G
B
Spc
L1
Bksp
C+PU
[{
-_
Tab
L2
F1
F2
F3
F4
F5
F6
F7
F8
F9
F10
F11
F12
Ins
L2
-_
]}
.>
1
F6
Y
6
F11
H
N
Ent
L1
2
F7
U
7
F12
J
M
RCtrl
3
F8
I
8
K
;:
Hme
Left
4
F9
O
9
L
PgUp
Up
PgDn
Down
5
F10
P
0
=+
End
Right
Bksp
Hme
Del
EndRAlt
Up
HUI
Left
SAD
Win+L
Enter
M_T
Right
SAI
Down
HUD
Vol+
VAI
Back
M_P
Mute
TOG
Vol-
VAD
Play
MOD

Leader key combinations:

+ W - Win
+ B - Reset/Programming
+ W,R - Command
+ W,L - Logout
+ W,S - Screenshot
+ W,T - Win+Tab (cycle Windows)
+ C,A,D - Ctrl+Alt+Del
+ C,A,E - Ctrl+Alt+End (for RDP)
+ A - Alt+Shift+Insert (Far Manager)
+ Q - Ctrl+PgUp (Far Manager)
+ S - Caps Lock

Compilation

Prepare your build environment by following QMK official Getting Started guide. Checkout code from my Github repository at https://github.com/kbjunky/IIICC. Default setup will compile with macropad support thus if it's not present (especially when OLED is not installed) it might cause some problems.

To **enable** macropad and OLED:
In **rules.mk:**

```
OLED_DRIVER_ENABLE = yes
```

In **config.h**:

```
#define MACROPAD_ENABLE
```

To **disable** macropad and OLED:
In **rules.mk:**

```
OLED_DRIVER_ENABLE = no
```

In **config.h**:

```
//#define MACROPAD_ENABLE
```

If you are having issues with the keyboard enable debug mode and check output either in QMK Toolbox or hid_listen.exe. Underglow needs to be disabled while console is enabled otherwise firmware will exceed allowed size:
To enable debug mode:
In **rules.mk:**

```
CONSOLE_ENABLE = yes
RGBLIGHT_ENABLE = no
```

You can also enable/disable boot animation and change it's interval.
In **keymap.c**:

```
#define BOOT_ANIM_ENABLED
#define BOOT_ANIM_DELAY 125
```

Hub can support a matrix of 3 devices of 8 columns and 8 rows each. Can be easily extended by modifying IIICC.h with a new key matrix.

Links

https://easyeda.com/pigboard - my electronics projects including IIICC
https://github.com/kbjunky/IIICC - IIICC Github page
https://www.coroflot.com/sean_yin - designer of ‘Hexa’ case art page
kbjunky#6476 - Discord

Enjoy your IIICC
Thank you!