RP2040 Development Board

30 September 2022

Abhinav Chinnusamy

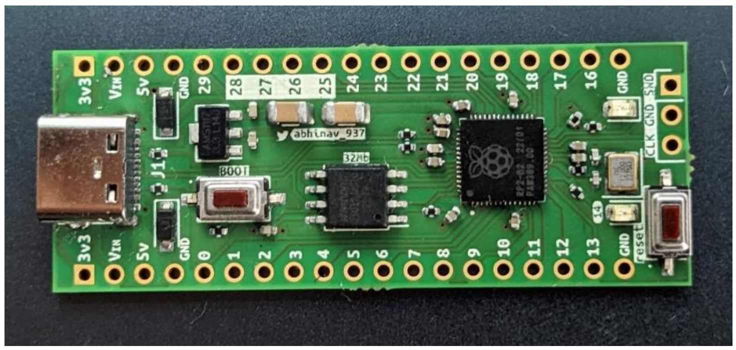


Figure 1. Top View of assembled PCB

Introduction

The RP2040 Development Board is a small, low-cost, versatile board. It has an RP2040 microcontroller, an onboard LED, a USB-C connector, and an SWD interface. The USB bootloader allows it to be flashed without any adapter in a drag-and-drop manner. It is also possible to flash and debug with its SWD interface using an external adapter.

Hardware

- Dual-core Arm Cortex-M0+ processor running up to 133MHz
- 264KB on-chip SRAM
- 32MB onboard QSPI flash with XIP capabilities
- 26 GPIO pins
- 3 Analog inputs
- 2 UART peripherals
- 2 SPI controllers
- 2 I2C controllers
- 16 PWM channels
- USB 1.1 controller (host/device)
- 8 Programmable I/O (PIO) for custom peripherals
- On-board LED

Physical Dimensions

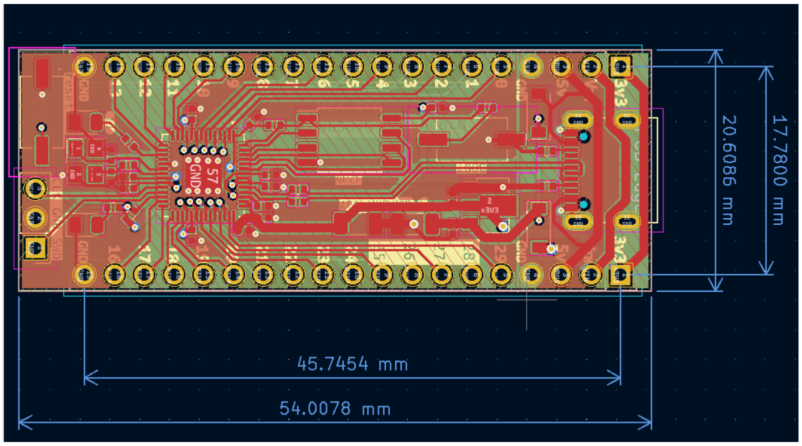


Figure 2.Board layout in KiCad PCB editor

This PCB was designed in KiCad 6.

The dimensions are 54mm*17.78mm.

Schematic

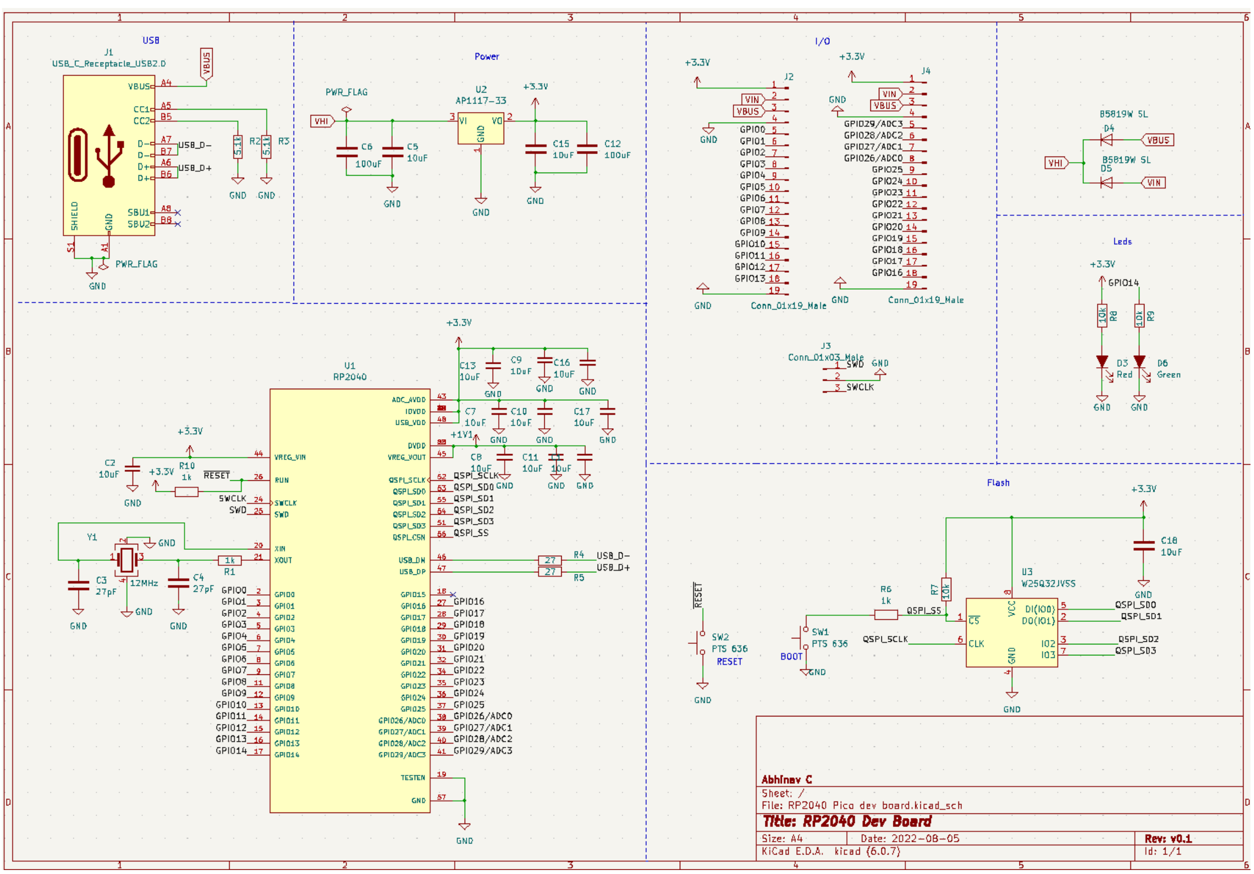


Figure 3. Schematic

Input supply

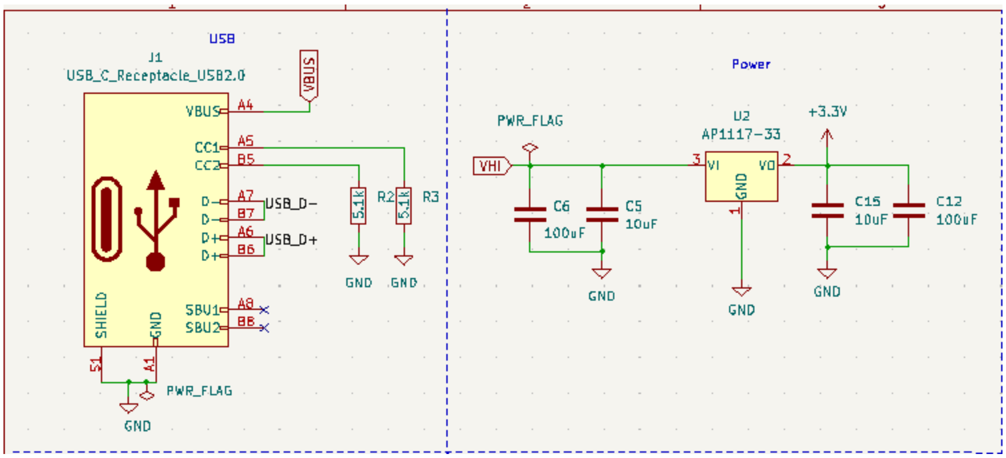


Figure 4. Power and USB Section

The input power connection for this Design is via the 5V VBUS of USB-C, as the CC pins are pulled down with 5.1k resistors. Also, it can be powered with the Vin pin up to 18V max and a min of 5V.

The Data lines in the USB-C port are connected to RP2040 directly, as it has native USB functionality.

We need only 3.3V for RP2040, so AMS1117 is chosen to deliver 3.3V up to 1A max.

It is advised to choose 0805 capacitors instead of smaller 0603 or 0402 because of DC Biasing.

Bigger the footprint, the lesser the DC Biasing (in MLCC, it also depends on brands). If the size is a constraint, more decoupling capacitors should be used.

Crystal Oscillator

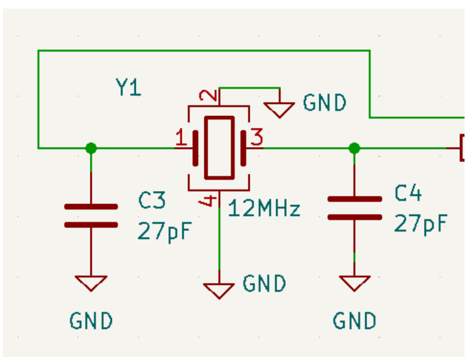


Figure 5. Oscillator

Do not forget to add load capacitance to the crystal. Otherwise, it might not oscillate.

Please choose the crystal's suitable capacitors to match its load capacitance. It's not a common value to use.

Refer to datasheets and calculate the C3 and C4(figure 5) for better results.

There are methods online to calculate it.

How many decoupling capacitors are to be used?

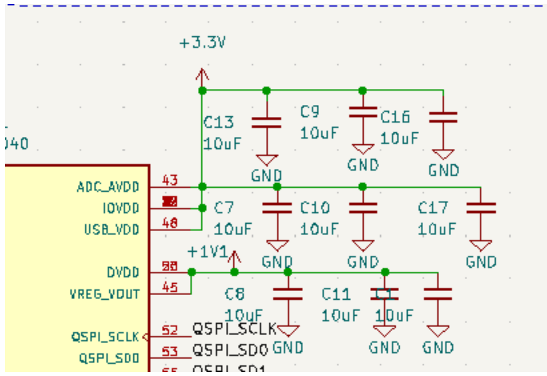


Figure 6. A section of the schematic showing the RP2040 power supply inputs, voltage regulator, and decoupling capacitors

For the power input of RP2040, we require some decoupling capacitors for filtering purposes.

Raspberry Pi suggests using at least seven 100nF 0402 MLCC caps on 3.3V Power and two 100nF, 1uF 0402 MLCCs on 1.1V power pins.

If more oversized (>0402) caps are used, fewer capacitors can be placed because of lower DC bias in larger footprint caps.

These capacitors should be placed very close to the Microcontroller as possible, as it prevents the voltage level in the immediate vicinity from dropping too much when the current demand suddenly increases.

LEDs

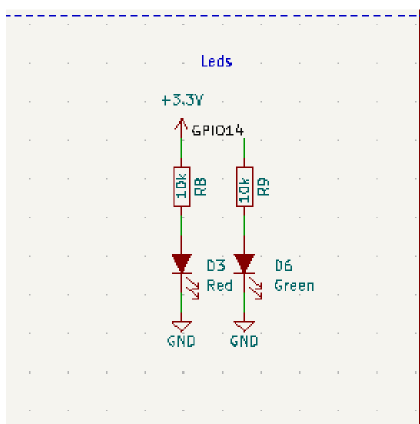


Figure 7. A section of the schematic showing the Onboard led and Power led.

There are two LEDs on the PCB.

One (Red) is connected to the 3V power track.

Other LED(Green) connected to GPIO pin 14 for Onboard LED.

PCB Design

The PCB was designed in KiCad v6 with 4 layers, the middle layers being gnd and the outer layers being mixed signal and Power.

Physical Stackup

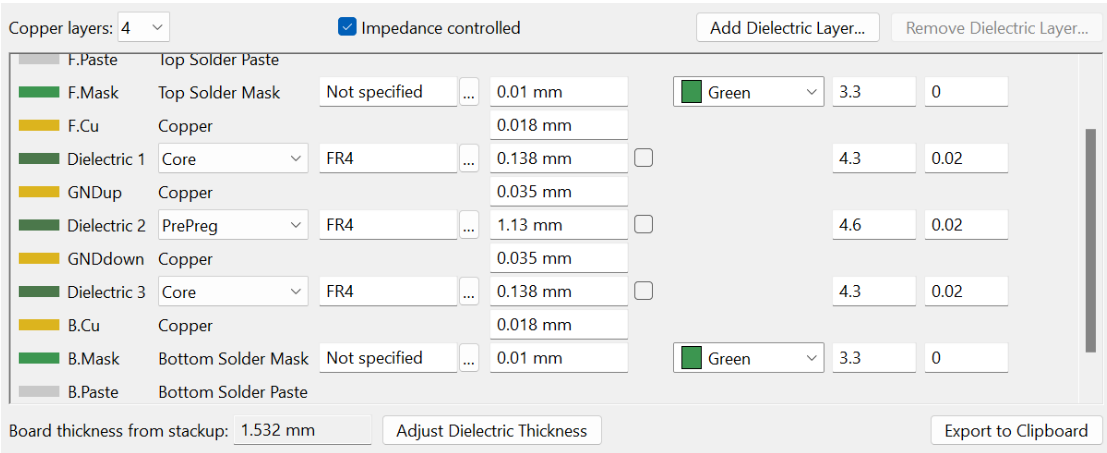


Figure 8. Stack-up specifications of the Board.

The above figure shows the stack-up specifications of the Board. This depends on the manufacturing capability of the PCB Manufacturer. I used the specs listed by AISLER for 4-layer Board.

Tracks and Vias

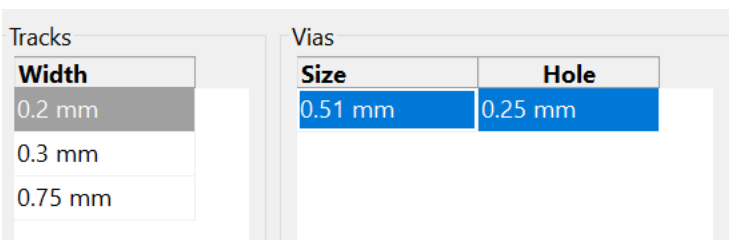


Figure 9. Pre-defined Track and Via dimensions.

I've used 3 different widths for tracks—the lesser width track(0.2mm) for connecting microcontroller pins to other components. And the broader track for Power.

Track width depends on the current it can handle under a specific temperature rise. KiCad has an inbuilt tool to calculate it.

One way to have narrow tracks is to increase the thickness of the copper layer. I used 1oz of copper thickness.

Increasing the thickness increases the manufacturing cost of PCBs.

Vias in uC's Copper Pad

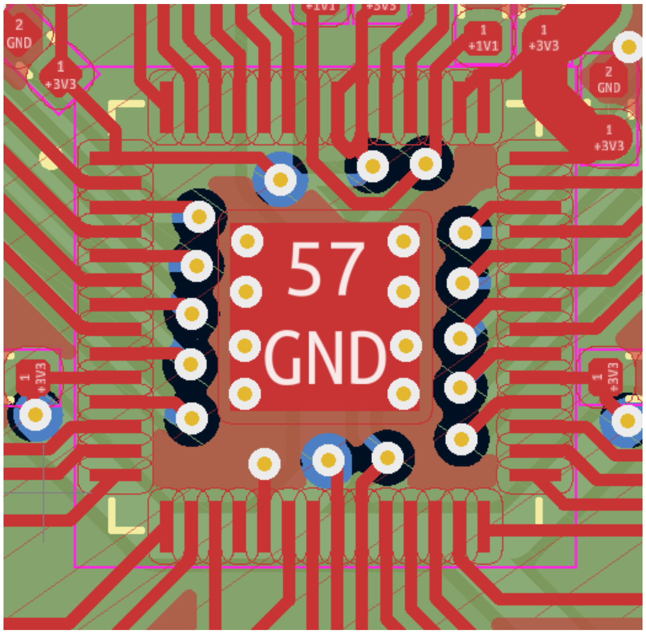


Figure 10. A section of the PCB Design showing vias placed in RP2040 footprint.

The unconnected vias on copper pads were placed for heat dissipation for the Microcontroller.

Copper fills

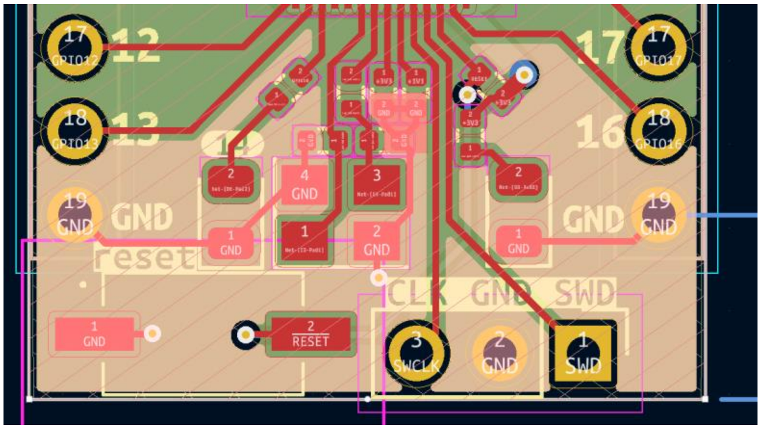


Figure 11. A section of the PCB Design showing the copper fill zone (highlighted).

Copper fill zones are placed in PCBs to increase EMI performance and heat dissipation, and signals can take the shortest path.

In this Design, I've placed copper fills in both inner layers, which are grounded. And there are a few copper fills on the top and bottom for Power and Gnd.

Breadboard Friendly?

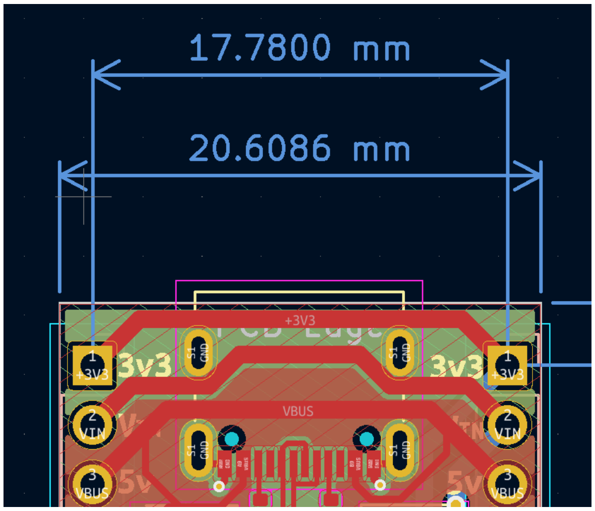


Figure 12. A section of the PCB Design showing distance between header pins pads.

This Board was made with consideration to be breadboard friendly. So, the header pins are separated at 17.78mm, like Arduino Nano.

Silkscreen Text

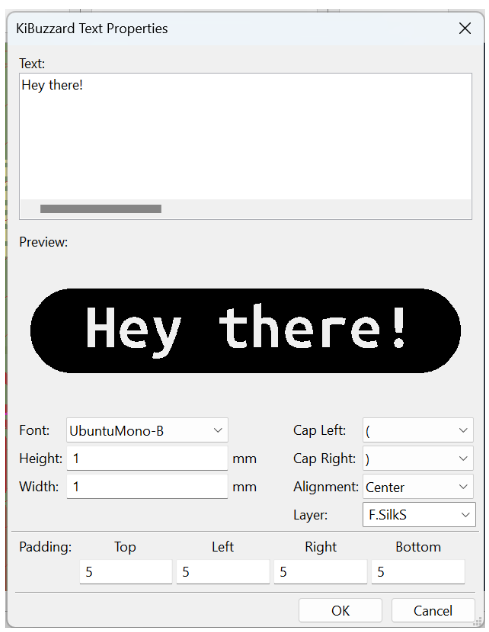


Figure 13. KiBuzzard tool for silkscreen text.

KiBuzzard is a KiCad Plugin for creating labels in various fonts and inverted backgrounds developed by gregdavill.

This plugin allows the creation of boxed texts in various formats in silkscreen. I recommend this plugin very much.

PCB Manufacturing

To manufacture the designed PCB, export the Gerber and drill files and upload them to the PCB manufacturer's website.

Or there are plugins for top-rated PCB manufacturers like AISLER, PCBWay, JLCPCB, et Cetra.

After installing the plugin, A single click to upload required files directly to their website.

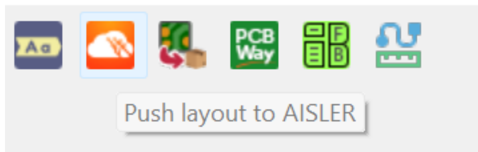


Figure 14. AISLER push plugin in KiCad.

It's preferred to order a stencil with the PCBs if assembly service is not chosen. There are 0402 components, and the Microcontroller lead pitch (pin to pin) is 0.4mm.

Personal preference:

I prefer AISLER for manufacturing the PCBs. There are no delivery charges for basic delivery service; they were delivered to India within 7 days after the order was dispatched, and no customs duties were levied.

Manufactured PCB from AISLER

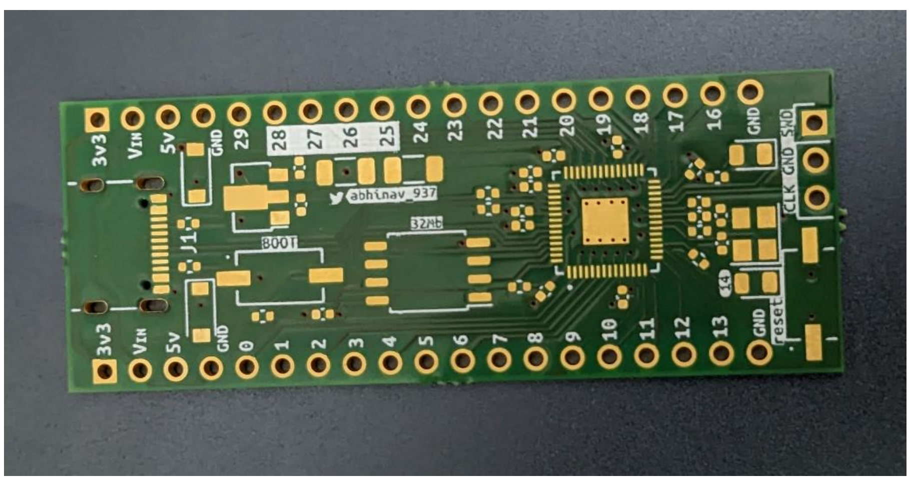


Figure 15. Manufactured PCB.

Component sourcing

I sourced all components from lcsc.com except RP2040. RP2040 was purchased from robu.in.

There were no custom duties levied on components. But it took around 15 days for delivery—longer queue times in customs clearance.

Purchase more components than what you require for assembly.

I lost many SMD components while placing them on Board.

Assembly

After sourcing all components and PCBs, tools like a Reflow oven or Hot Plate, Solder paste, Flux, and soldering iron are required.

I reflowed the components with a Reflow Oven (Model: T962A).

Figure 16. T962A Reflow oven

Assembly process:

1. Solder Paste deposition
2. Parts placement
3. Reflowing the PCB
4. Rework on bridged pads

Solder Paste deposition

To apply solder paste on the pads, Secure the PCBs on a flat surface with pre-cut holders. If you don't have one, you may use some trash PCBs of the same thickness to hold it.

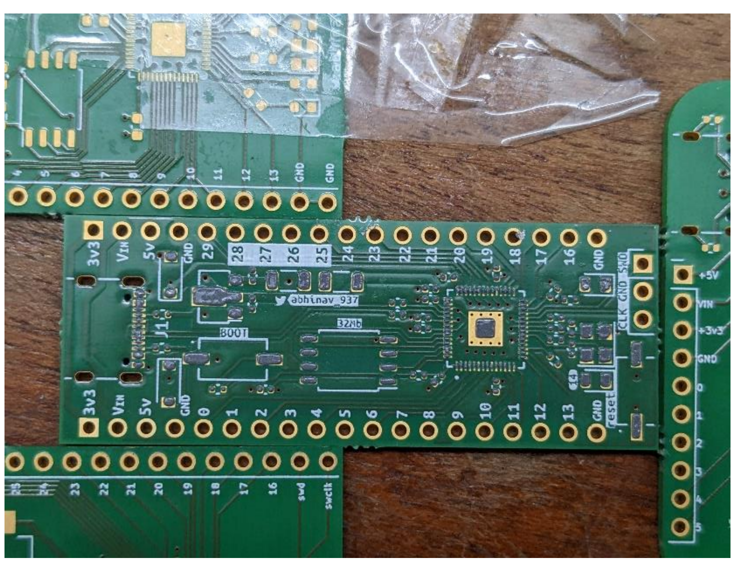


Figure 17. Securing the PCB with trash PCBs on a flat surface.

Then align the stencil on the top of the PCB, ensure it's properly aligned, and secure it for best results.

Figure 18. Aligning the PCB and securing it.

Prepare the solder paste by mixing it properly. Make sure to check that it's not expired.

Refrigerated solder paste gives the best results.

Take some amount(excess) solder paste and place it on one side of the stencil.

Using a squeegee or an old credit card, slide the solder paste on the stencil at an angle of 60˚.

Then gently remove the stencil.

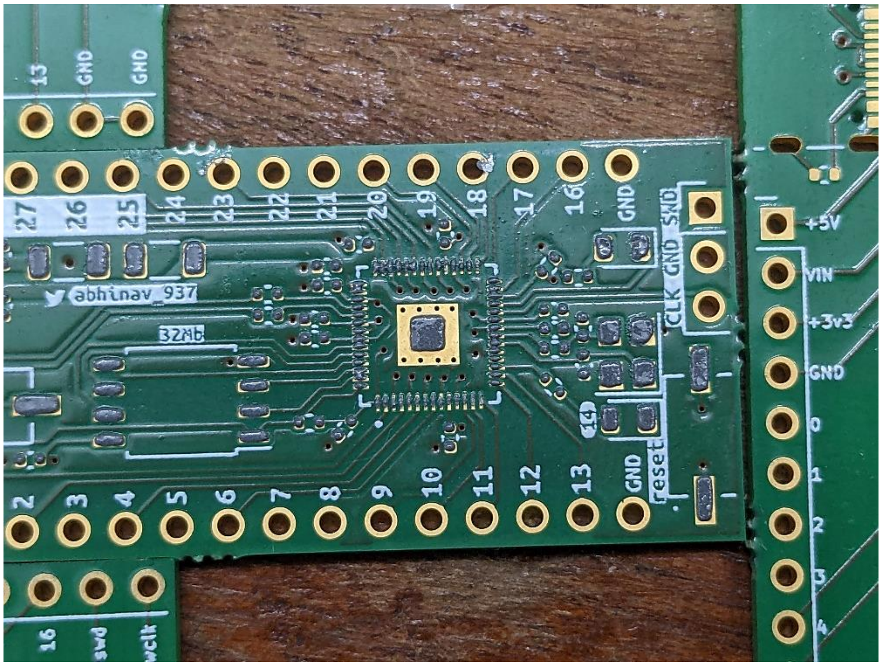
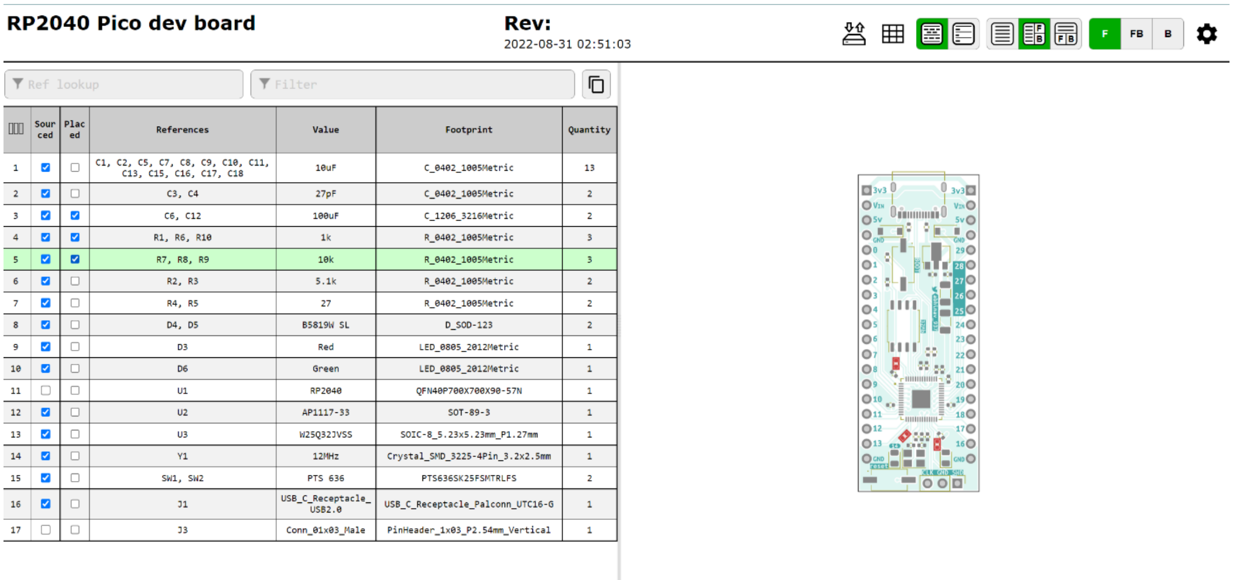


Figure 19. Solder paste applied PCB.

Parts placement

Before placing the parts, have a copy of BOM to mark the components placed and sourced to avoid confusion during placement.

I prefer iBOM, a KiCad plugin, and a convenient BOM listing that can correlate visually and easily search for components and their placements on the PCB.



	Sourced	Placed	References	Value	Footprint	Quantity
1	☑	☐	C1, C2, C5, C7, C8, C9, C10, C11, C13, C15, C16, C17, C18	10uF	C_0402_1005Metric	13
2	☑	☐	C3, C4	27pF	C_0402_1005Metric	2
3	☑	☑	C6, C12	100uF	C_1206_3216Metric	2
4	☑	☑	R1, R6, R10	1k	R_0402_1005Metric	3
5	☑	☑	R7, R8, R9	10k	R_0402_1005Metric	3
6	☑	☐	R2, R3	5.1k	R_0402_1005Metric	2
7	☑	☐	R4, R5	27	R_0402_1005Metric	2
8	☑	☐	D4, D5	B5819W SL	D_SOD-123	2
9	☑	☐	D3	Red	LED_0805_2012Metric	1
10	☑	☐	D6	Green	LED_0805_2012Metric	1
11	☐	☐	U1	RP2040	QFN40P700X700X90-57N	1
12	☑	☐	U2	AP1117-33	SOT-89-3	1
13	☑	☐	U3	W25Q32JVSS	SOIC-8_5.23x5.23mm_P1.27mm	1
14	☑	☐	Y1	12MHz	Crystal_SMD_3225-4Pin_3.2x2.5mm	1
15	☑	☐	SW1, SW2	PTS 636	PTS636SK25FSMTRLFS	2
16	☑	☐	J1	USB_C_Receptacle_USB2.0	USB_C_Receptacle_Palconn_UTC16-G	1
17	☐	☐	J3	Conn_01x03_Male	PinHeader_1x03_P2.54mm_Vertical	1

Figure 20. iBOM tool.

To place the SMD components, use a bent-tip tweezer to pick and place it on PCB and give a gentle push from the top.

Try to place the components properly on the first try. Re-positioning may push the solder paste and disturb it.

Do not place THT components for reflow. It can be hand soldered later.

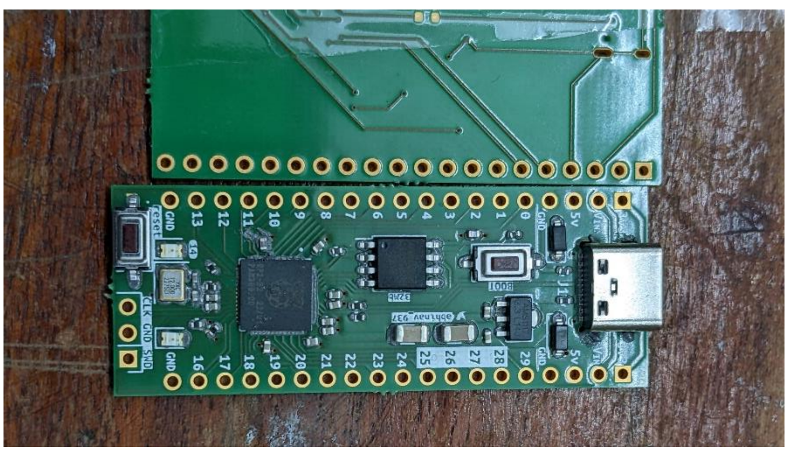


Figure 21. PCB after placing the components and ready for reflowing.

Reflowing the PCB

The first step in reflowing is to set the temperature profile in the reflow oven.

T962A reflow oven has 8 different temperature profiles. You can choose one after referring to your solder paste datasheet and components datasheets.

I used a custom profile which gave a better result for the XG-50 solder paste.

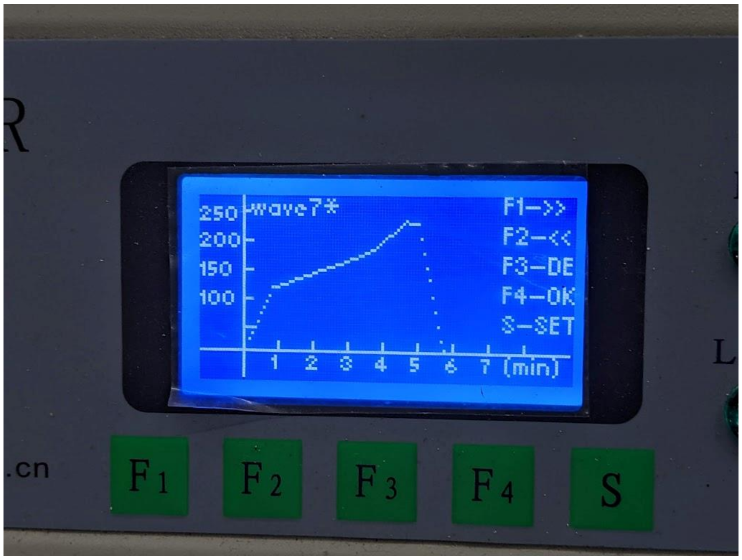


Figure 22. Custom reflow profile on T962A Oven.

Now, gently place the PCB in the Oven and start the reflow oven.

Using this Oven in a ventilated room or with a venting system to extract the fumes is good practice.

Figure 23. Semi-assembled PCB placed in Oven for reflowing.

It takes 5 minutes and 20 seconds to complete the reflow for this solder profile. When the Oven starts beeping, please turn it off and wait some time for cooldown as the inside temperature could be more than 100˚.

Please do not take the Board right after the reflow process ends; the components are loose in hot solder.

After the Board cools down, take it out and check for solder bridges and short circuits.

Rework on the bridged pads

It's not always that we get a perfect reflow.

Due to excess solder paste on some pads, the neighbouring pads get bridged.

The uC in my PCB got bridged pins.

So, we can rework it using one of the following methods.

1. Hot air rework station

We can reflow the component using the hot air station and scrap the excess solder.

Use Kapton tape to cover the other components while reworking on the uC.

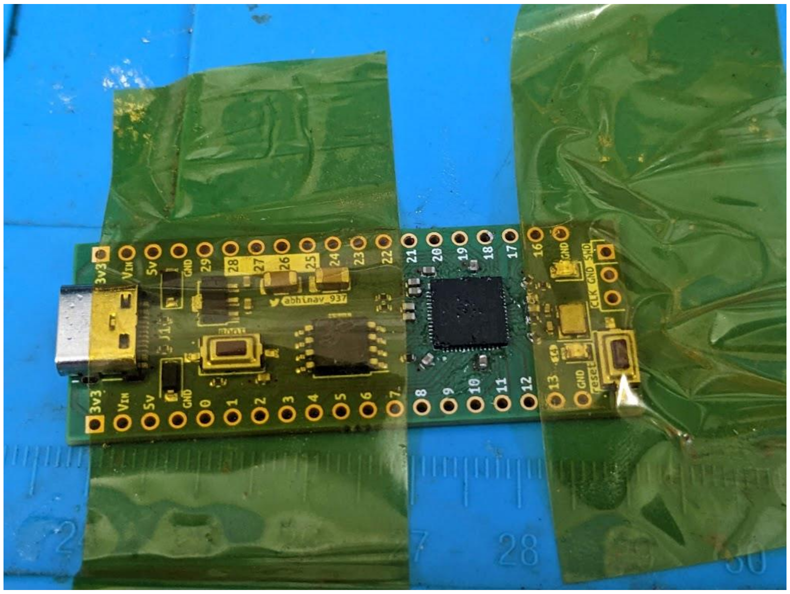


Figure 24. Assembled PCB Kapton taped for rework on the uC.

This method was not successful for me in getting rid of bridged pads.

2. Using a soldering iron

My idea was to try using a sharp tip soldering iron to remove the bridges, but I was unsuccessful.

Next, I tried using a properly tinned, angled flat tip soldering iron and successfully removed the bridges.

But still, there were shorts in the Power and GPIO pins of uC.

Finally, I came to know that uC was bridged from the inside due to excess solder paste on the pads.

So, I tried another method to resolve it.

3. Reflowing once again in Oven

I removed the uC from the PCB using the Hot air station and cleaned the pads and added some solder to it.

Then I applied some flux to the PCB, placed the uC, and reflowed in Oven.

Now there are no bridged pads, and connectivity is fine.

Still, the pads and vertical side of uC were not soldered properly. So, I reworked it with a soldering iron.

NOTE:

Use an ample amount of Flux while doing any rework.

Reflowing the PCBA multiple times can change the properties of the components.

E.g., Reflowing the crystal oscillator more than once can change its frequency.

After resolving all the issues and checking the connectivity of pads and power tracks, we can solder the THT components and Power up the Board.

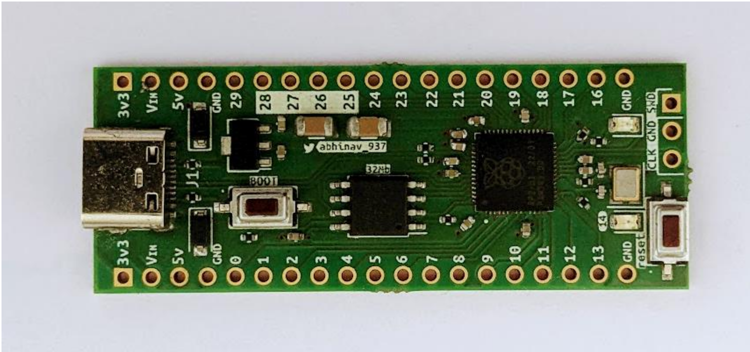


Figure 25. Fully assembled working PCB of RP2040 dev board.

Testing

Ensure the Board has no shorts, and connect a USB-C cable from a charging brick or a computer.

If the Onboard Power led lights up, then Board is working fine (At least in the power section).

Connect the Board to a computer while pressing the Boot button and release it when mass storage (RPI-RP2) is detected.

This ensures that Board is working with full functionality.

If mass storage is not detected, solder the Flash pins of uC to pads properly or check for bridging.

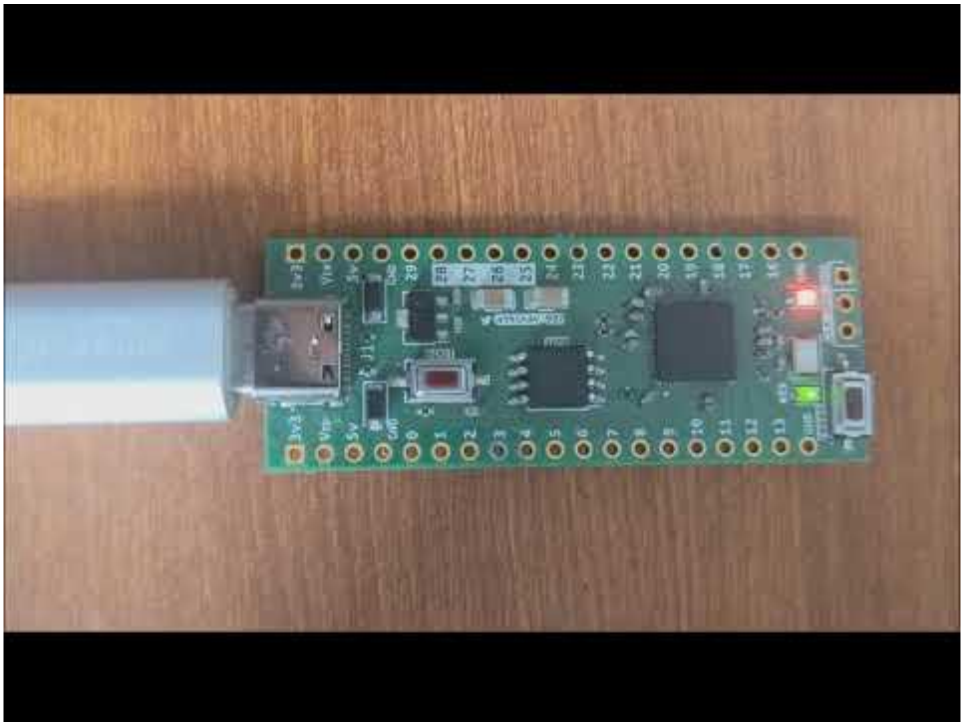
NOTE: No drivers are required for USB serial connection.

Once Everything is working correctly, we can drop the compiled .UF2 file on the mass storage, and it starts running the program.

RP2040 officially supports C/C++ and MicroPython.

For MicroPython, Thonny IDE allows you to code in MicroPython, compile, and run on the Board by selecting the proper interpreter and flashing a special firmware.

For C/C++, VS Code and a few extensions allow us to write and compile the program with debugging functionalities.



On-board Led Blink test

References

https://datasheets.raspberrypi.com/rp2040/hardware-design-with-rp2040.pdf

https://twitter.com/sad_electronics

https://www.youtube.com/watch?v=kcwvuwetgEQ

https://www.digikey.in/en/maker/projects/hardware-design-with-the-rp2040-part-1-schematic/c4326f0fd813413698d617cf625125ee

https://github.com/mrangen/RP2040-Dev-Board

https://github.com/gregdavill/KiBuzzard

https://www.kicad.org/

https://aisler.net/

https://lcsc.com/

https://www.puhuit.com/main/page_products_t962a_ir_ovenic_heater.html

https://thonny.org/

A Special thanks to sad_electronics and peg_iitdh for their continuous support.

Design files: https://github.com/abhinav937/RP2040-Dev-Board