Microbox 2K2 Construction guide

Introduction

This document contains some brief notes to assist during the assembly of the MB2K2 PCB. It assumes that the reader has some experience with fine pitch SMD soldering, if not don't panic as there are any number of online resources and YouTube videos that go over the basics.

As with all SMD assembly it's important to follow good safety procedures particularly to ensure that you have good protection from the fumes generated from soldering and from any hot solder splashes, again there is a wealth of online information concerning protection from these and other hazards, follow it!

The tips and suggestions below are intended to be guidelines and don't need to be followed slavishly, if you have a better way of doing things then by all means use it.

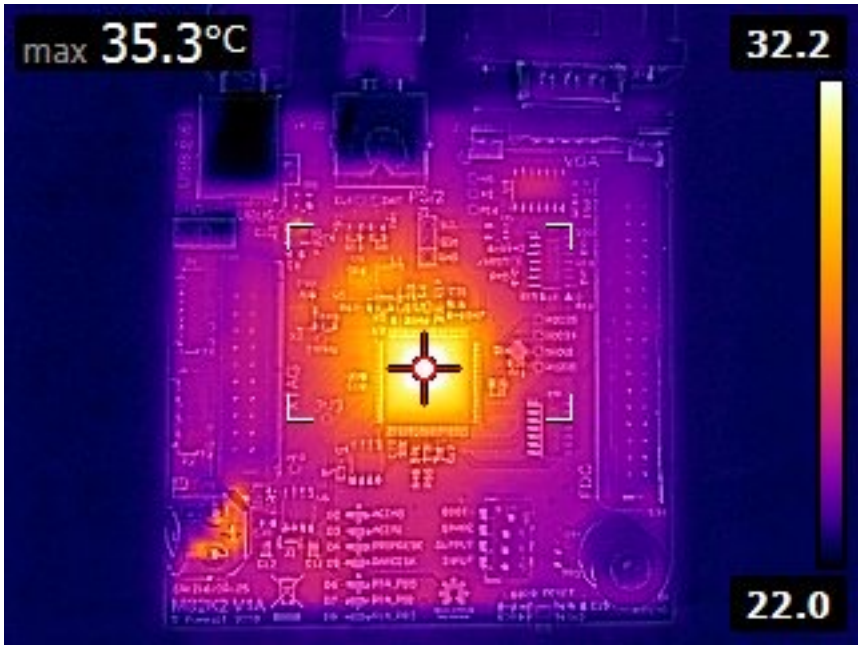
The PCB has been designed with an eye towards manual assembly and has a number of features to simplify the process. It's entirely possible to assemble the board using just a temperature controlled soldering iron and without needing to use a hot air gun or reflow plate.

BOM

The component BOM is given in the hardware folder of the release package. For convenience it's also accessible as a shared 'live' BOM on Octopart. At the time of writing all the components are really available and the total BOM cost is < $50.

Construction notes

- R51 is no fit
- R55-57 are optional components
- The surface finish of the PCB is ENIG as this eases the soldering of the fine pitch components. With this finish it's important to clean the PCB thoroughly before assembly and to use a good quality flux. I've found that one of the no clean 'gel' fluxes gives the best results.
- Assemble the 3.3V and 1.0V power power supply components first and test that these are working correctly before proceeding with the rest of the assembly. This can save some expensive problems later on. The switch mode regulator chip U2 has a 'hidden' ground pad underneath its package. So that this device can be attached without reflow there is a large via in the PCB underneath the parts and this allows the hidden pad to be soldered from the bottom of the PCB. This feature also has the side effect of forming a good heat sink to the inner layers of the PCB although in practice this isn't needed as the design runs very cool in use as this thermograph shows:-

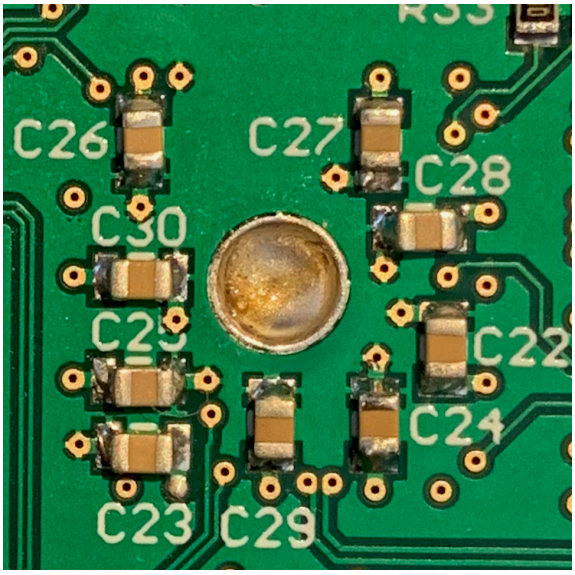


- The XU216 should be added next, this is probably the most difficult part. Be sure to leave the XU216 in its moisture sealed packaging until just before assembly as the part is prone to 'popcorning' if left in a humid atmosphere too long. The leads of this part are quite fragile, if possible handle just with a vacuum pencil to avoid touching the leads.

Use plenty of flux; this is where the gel flux is handy because it holds the part in place but still allows fine adjustments of position. (Don't forget to put some flux on the ground pad in and around the via). Check that pin 1 is in the correct place! (there is a flat corner and depression marking on the part and a dot on the PCB to show the location of pin one). When the part is correctly positioned, tack down one lead near a corner and check the alignment again, if ok tack a pin on the opposite corner.

Because of the fine pitch of the leads on the part, the best way to solder the leads isn't to use the traditional technique of applying the solder to the joint, instead it's better to apply a small amount of solder to the iron tip and apply this to the very end of the pin allowing the solder to wick along the pad to form the joint. The PCB features solder mask slivers between the pads (this would not normally be a good idea in a mass production PCB) which helps discourages the solder from bridging between pins and if any solder bridges do form the solder mask assists clearing these with solder wick.

In the same manner as the switch mode regulator the XU216 has a hidden pad underneath the device which is its only connection to ground. This pad carries all of the logic and I/O current so it's important to ensure that a good connection is made. As before there is a large via which is used to allow access to the ground pad from the rear of the PCB so when the signal leads are soldered then the ground connection can be made. Use plenty of flux here and flood solder into the via. Heat will tend to flow into the internal planes of the PCB so use a hot iron and beware of forming a 'cold' solder joint. When complete it should look something like this :-



- After the XU216 is in place, place the other SMD components. For the discrete parts flux both pads then tin one of the pads. Then slide the part into place with fine point tweezers whilst melting the solder to make the first joint and then solder the other end. In practice this takes less time to do than to describe. I find that it's easiest to work down the BOM placing all the instances of a particular component value in one pass.

- After placing the SMD parts place the through hole parts last of all and give the PCB a good clean with IPA and an old toothbrush.

Initial testing

- Briefly apply power to the board and check current draw, it should be in the region of 150mA, much greater than this indicates a short somewhere or a reversed IC. If the initial 'smoke test' is OK check that the 3.3V and 1V power rails are correct.

- If a scope is available check that the 24MHz clock is present and that reset is asserted following power up.

There isn't much else that can be checked at this stage so the next step is to flash the firmware as detailed in the 'Getting started' guide.