Frequency Counter kit

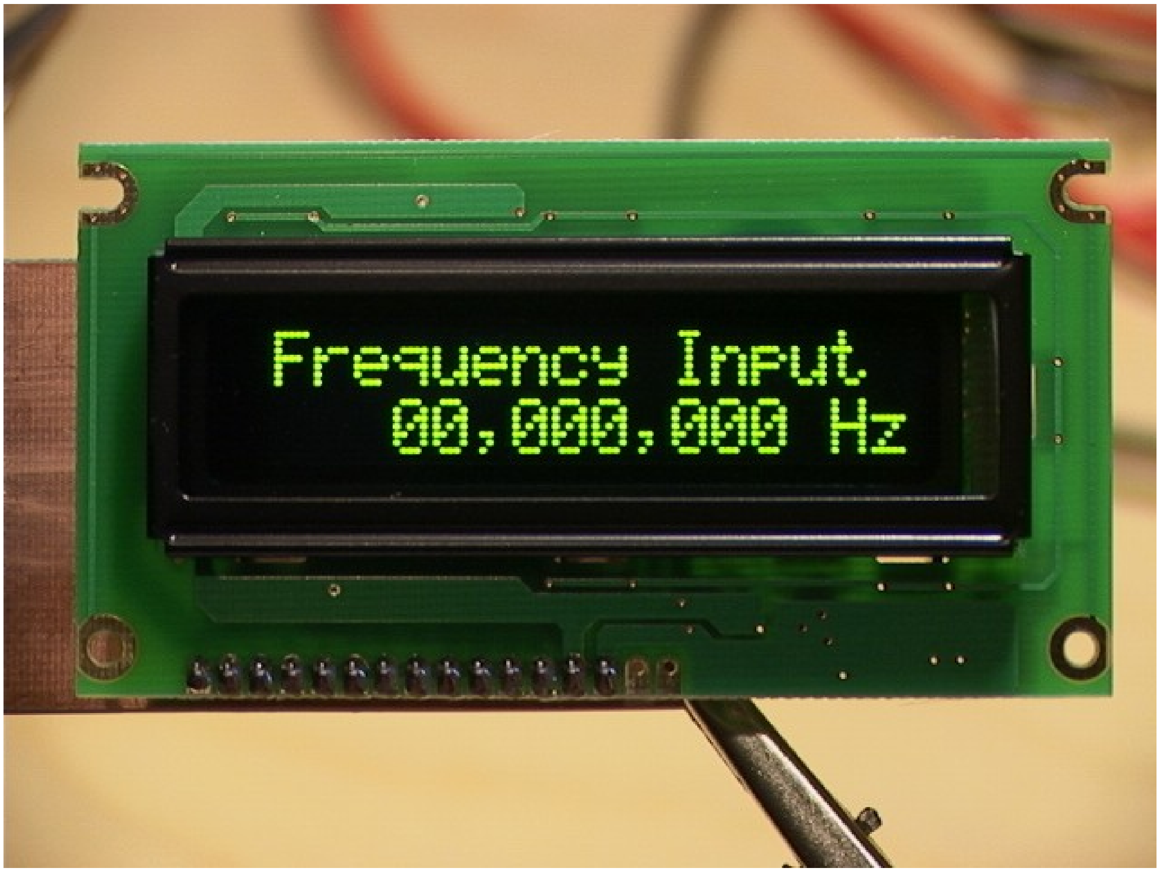

Illustration 1: The frequency counter, driving a 2x16 OLED display

Table of contents

Building the Frequency Counter

Preparations & tools

Fixing / holding tools

I used two helping hands, one of them equipped with a magnifying glass. These are really helpful in many situations. Placing the pcb flat on the bench is not recommended as it will move around when you poke at it with the soldering iron and if you place the palm of your hand on the bench (as you need to do unless you have rock-steady hands) the pcb might stick to your hand and lift when you lift your hand. When it drops back to the table tacked components might come off and be lost among the dust on the floor.

Illustration 2: Helping Hands

Why two of helping hands? Well, the molded base of the helping hand could be a little heavier. It has a tendency to tip when the pcb is mounted and the center of gravity is a little off the center of the base. This little problem gets worse as you connect your scope and whatnot to the pcb to take measurements. For this reason I like to use two of them, one on each side. Sometimes I use only one, it depends on the situation. I am not really happy with the magnifying glass. I'd rather use one of those large illuminated ones that have their own bench mount with arm, much like my lamp. When I come across one of these at a reasonable price I'll definitely get one.

Pliers, tweezers and scrapers

I used three different pliers. Two are for cutting and one is for bending and holding.

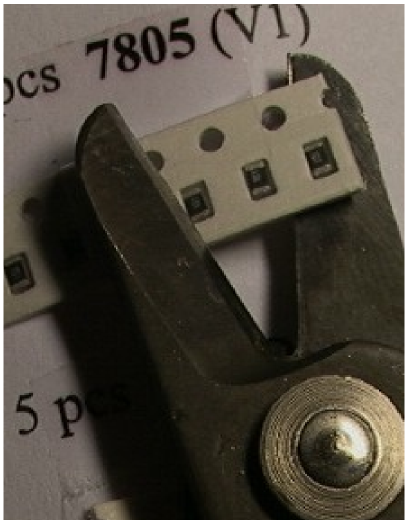

Illustration 3: Cut components as you need them to prevent loosing any of them

The straight cutter I used mainly for cutting the components one by one as I needed them from the tape that came taped to a sheet explaining what each of them were. If you remove e.g. all 0603 size 100nF capacitors and place them on your desk and then sneeze they will fly all over your room and you will never find them. They are really tiny! Be careful so you don't loose any components. A recently cleaned floor helps in finding components you accidentally dropped.
The angle-cutter I use for cutting component leads and vias flush with the solder on the pcb and also for cutting component leads and pieces of wire to length.
The bend / hold pliers is nice when bending small pieces of wire for the vias. You can use your fingers as well.
I used two sizes of tweezers, one with a very fine tip for placing and holding 0603 sized components and a standard size tweezer when I needed a little more holding force.


Illustration 4: Pliers, Tweezers, Scrapers, solder-braids, Flux-pen & solder

I use a scraping tool, similar to what the dentists use to rid your teeth of calcium buildup. This is nice to use for following the “tracks” between the pads and the ground-plane. This way you can feel any tiny bridges or dirt stuck in there. This actually helped me find the physical location of the only problem I encountered when building this kit.

Soldering tools

I have a Weller soldering-station and two irons for it. A smallish one and a standard size one. I only used the small one for this project. When soldering components I had the temperature set to just over 300° centigrade. When soldering the vias I set it to just under 400° as it takes a little more energy to heat the backplane because there are no thermals. Generally the more metal you need to heat a higher temperature will be needed. I know I have the right setting when I don’t need to heat for more than a few seconds for the solder to flow nicely. I mostly used the 0.32mm included solder except for the vias where I used a 0.8mm solder.

Measuring equipment

I am spoiled and have a Fluke ScopeMeter. This is a combined digital oscilloscope and multimeter. It has interfacing (optical RS232) so you can connect it to your computer. You will mostly be OK with a multimeter, bench or hand-held model. If you build and play a lot with electronics you should get some sort of scope. There are USB scopes that use the pcto display values and sometimes you can get cheap scopes on ebay. Sometimes analogue scopes can show things (glitches) that the digital ones have missed. I have no examples of this but I do remember reading an article on the net where someone troubleshot for ages and only found the problem when switching from a digital scope to check with an analogue one.

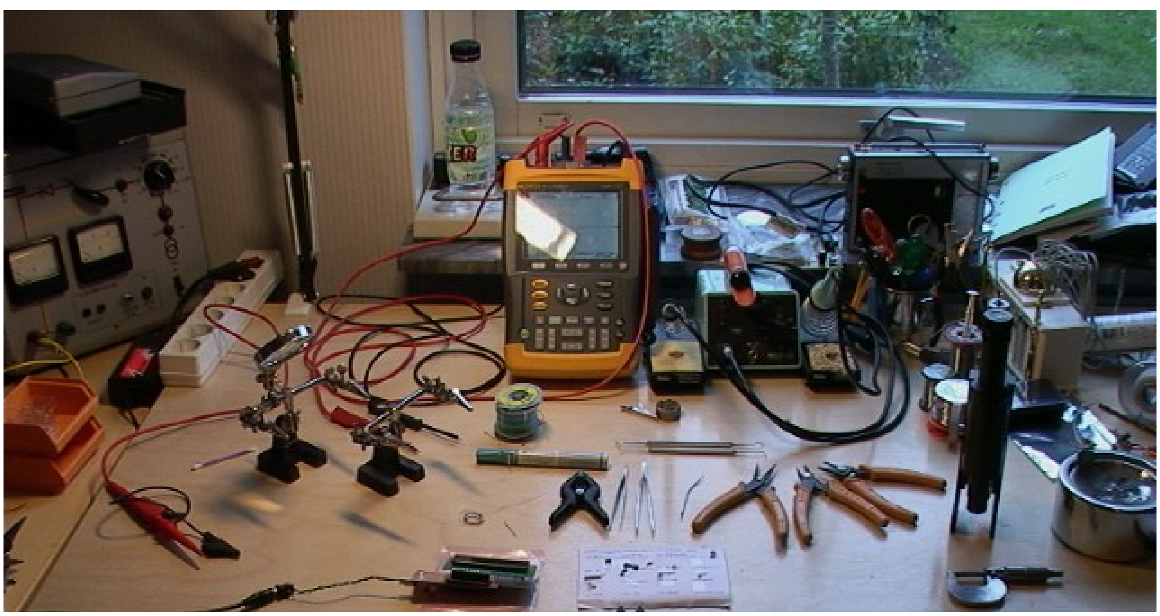

Illustration 5: My workbench as set up for building this project

I also used a microscope to verify optically that I did not have any solder-bridges and that all leads on the LMX2322 were nicely soldered. Use it if you have it.

Workbench / table

Clean up your desk and remove any tools you don't plan on using. This helps you find what you need when you need it. I hate having to spend time looking for things when I’m in the middle of something. It brakes my concentration.
Good lighting is a must when you work with tiny components such as those used in this kit.

Plans / papers

Print and make sure you have handy any plans and layout documents you might need.

Building

OK, enough already with the preparations and tools. You should have a neat looking table / bench with the soldering tools untangled and the components within reach, the helping hands in front of you with the pcb clamped and everything well-lit.

I started with soldering the LMX2322 PLL as it was the trickiest component to solder. Also because it would be easier to inspect the soldering with the microscope with only the PLL mounted.

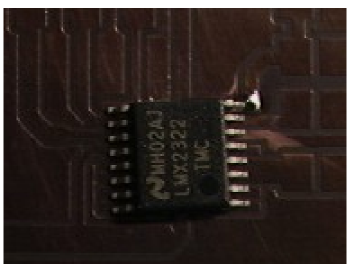

Illustration 6: Tack-soldered PLL

I placed a very small amount of the included solder on the top right pad, just enough to wet the area where the lead of the PLL goes. Now it’s time to place the PLL.
Make sure you orient it correctly. Look at the pcb layout, check for the dot that indicates pin #1. when the leads of the PLL are all aligned over the corresponding pads on the pcb I melted the solder I had previously added to the pcb-pad so it fixed the PLL to the PCB. This technique is sometimes called tack-soldering. The PLL might not be perfectly placed but as it is only soldered in one place it can be moved a little. So move it a little until it is perfectly aligned, be careful so you don't bend the tacked lead too much. When alignment was perfect I added a small clamp to the far half of the body of the PLL. It is now fixed *and* held flat to the pcb so we can solder a few more leads on the same side. Use very thin solder as this helps with not getting solder bridges. If you do get them they can be removed with a solder-braid later. I do not much like to use solder-braid (need some practice I suppose) as you need to heat them a lot. When a couple more leads have been soldered I moved the clamp to the other half of the PLL’s casing so I could solder the side that was covered by the clamp. Solder a few leads and then remove the clamp. Now solder the rest of the leads. When I was done I inspected the work using a microscope. Some leads had too little solder, I added some more. I managed to solder the whole thing without any bridges. If you do use a solder-braid use a very small one. They come in different widths.

Illustration 7: Clamped and fixed PLL

I measured the resistance between ground and between most of the leads, they all had resistance in the MΩ range or infinite resistance. Except for the ground pins, of course. I did this to make sure there were no bridges I could not see with the microscope.
I found that now was a good time to take a break and have some coffee as this little part of the project strained eyes, hands and concentration. Recharge your batteries. It is also a good thing to leave the project for a little bit. At least for me as I become like a kid on Christmas, well excited and in a hurry to get it all done. Don't hurry! Take your time to reflect and think your next steps through thoroughly. Your project will come out all the better for having done that.

For the next step I choose to solder the capacitors and the 100Ω resistor that make up the RF front-end. I drilled holes for extra vias around here. Do not solder R5, the 100KΩ resistor, not having this resistor will let the PLL self-oscillate when testing the circuit later. R5 was the very last part I soldered and only *after* testing and troubleshooting. I marked the plans and layout, circling this resistor and writing "LAST" next to it so I would not forget and solder before testing. De-soldering SMD components can be a little tricky and if you use force to get them off sometimes the pcb-pads come off as well if you're not careful.

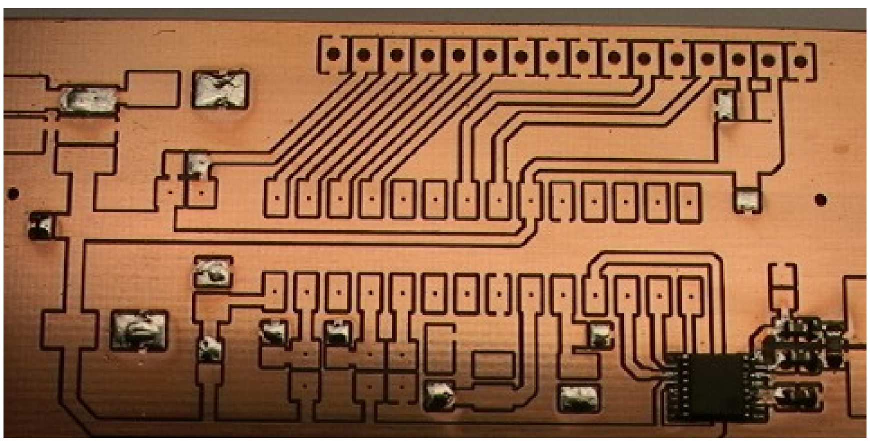

Illustration 8: RF-FrontEnd done and tack-solder added to pads

When done with that part (I like to think of that part as the RF part) I added a small amount of solder to one pad for every SMD component on the PCB. I then soldered two or three of them at a time by tack-soldering them to the pcb a few at a time, then soldering the non-tacked side neatly and then re-soldering the tacked side. The trim-pot has a pad under the body of the component. As this pad is also connected to the three-pad terminal I did not try to solder the pad under the trim-pot. You would probably need hot-air tools to get this done nicely so skip it if you do not have hot-air tools or some other way of getting it done.

When I had soldered everything except for the PIC and the Pin-header for the LCD I measured resistance to make sure there were no bridges that would short-circuit the 5V output of the 7805. When satisfied I connected power to the input of the 7805 and measured the output voltage of the 7805. It was a nice steady 4.990 Volts. So, all was OK so far.

Soldering the vias

I drilled some extra holes for some extra vias. I don't think you can ever have too many ground-vias. They help with shielding and leading of excess heat. Good grounding is extremely important when working with very high frequencies. Rip open a cellular phone and have a look. Also look at the PCB design-tips that can be found at the end of most data sheets for PLL's and other components working at several Gigahertz so you get an idea.

It takes a fairly long time to heat the copper on the ground-planes because there are no thermals. Having a larger mass that is hot helps. This means having a big blob of molten solder on the tip of your soldering iron helps heating the copper. The copper can't lead all that heat away very fast so it heats up and you get done soldering the via quicker with less risk of having a cold solder-joint. I set my iron to just under 400°. Using a larger soldering-iron also helps. I used the small one mostly because I'm a little lazy and changing irons plus waiting for the iron to reach working temperature just doesn't appeal.

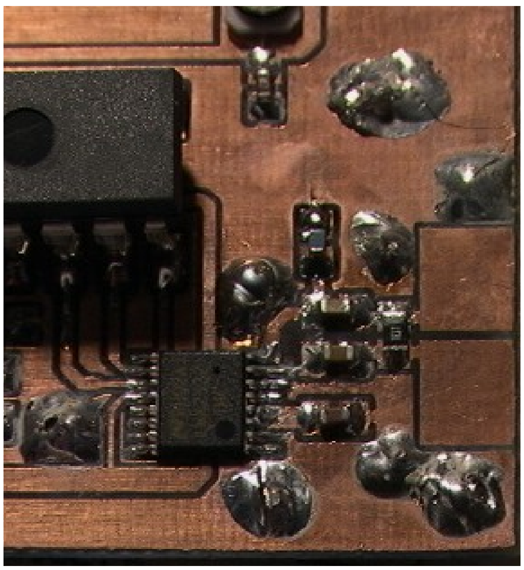

Illustration 9: I added seven extra vias around the RF FrontEnd. There's one under the PIC too.

For the vias you can use component leads, wire or whatever you have lying around. I used hook-up wire with a single lead. I bent the end of the wire 90°, the bent bit being a few millimeters long. I then inserted the bent bit through the hole and placed the pcb flat on my bench with the bent inserted bit facing upwards and the rest of the wire under the pcb. Add the solder-blob to where the wire and the copper meets and heat a little so the solder flows and wets the copper nicely all around the protruding wire. When satisfied flip the pcb over, straighten the wire, cut it so it protrudes a few millimeters and solder the side now facing you using the same technique. When done cut any excess wire so the solder-joint is fairly flush with the pcb. If there's a lot of solder you can cut that too. Do this for all vias. When soldering vias close to ground pins of components be careful with the heat as it will transfer to the ground leads of the component and heat it. Components do not like to be heated to 400°. If designing pcb's add thermals for the vias, this really helps you solder them without heating and cold-solder issues.

Soldering the PIC

Time to solder the PIC, I found this a little tricky because it was hard to heat the very fine tips of the PIC's leads and the PCB pads at the same time. To fix the PIC to the PCB I used the same technique as discussed earlier: tack-solder one lead, align, tack-solder another lead, then solder the rest of the leads, then go back and re-solder the leads you tacked first.

I found that heating the pads and adding solder to them a split second before touching the PIC's leads with the soldering-iron was the right way about it. If you start with heating

the lead the solder tends to stick to the lead and getting it to flow to the pad was tricky. Be careful so you don't overheat the PIC.
OK, the PIC was soldered. No problems so far. I soldered the pin-header for the LCD.
I measured resistance once again and found no bridges that would short the 7805. The 7805 has internal protection so it would probably not be a problem but I think it is better to be safe than sorry. Measuring and checking every now and then helps you find problems shortly after they came into the picture. If you solder everything in one go and then measure and find strange things it is a lot harder to figure out where it went wrong. So step by step, check up on what you just did. If all is OK then move on to the next step. Working methodically really helps. Take breaks when you feel tired or frustrated, or when you need some coffee. Don't eat and munch where you work, take a few minutes to down some sandwiches or something.

Testing and troubleshooting

All components are mounted (except for the 100KΩ resistor). I soldered a sort of polarized pin-header to the input of the 7805. I also drilled the ground input pad so that the pin-header became a via between the ground-planes. I like vias.
I connected power to the unit (I used 12 volts), measured the output of the 7805 and inputs to PIC, PLL and LCD, they were all 5 volts. Everything looks OK. I disconnected power and then connected the LCD. I soldered female mating headers to the LCD so I could remove it without having to de-solder it. I like connectors too.
When powered up the LCD displayed nothing at all. This is OK as you need to set the contrast with the little trim-pot. Adjust the pot, preferably with a non-conducting trimming tool so you won't short-circuit anything if you slip. I did this and got some output on the LCD but only square blocks. No text. This means something is wrong. I had expected the PLL to self oscillate and the PIC to output this frequency to the LCD. But I got nothing. I disconnected power, realized I was frustrated so I had a break. Ever heard of Murphy and his law?

During my break I thought things through while eating a ham and cheese sandwich. There are 5 volts feeding all components. The display outputs something. So the problem was probably communication between the PIC and the LCD somehow. I started measuring the PIC. There were 0.2Ω resistance between pin 9 (clock input) and ground. This is bad. I measured pin 9 with my scope. This should have said 13 Megahertz as it is the clock-pin. There was no oscillation. If the PIC has no clock-signal it will not run its program. I had found a problem. Looking at the components and the pcb layout I thought that the trim-capacitor could be the issue. I de-soldered it, very carefully, and took new measurements. The short to ground was still there. I still had 0.2Ω resistance between ground and pin 9. 0.2Ω is the resistance in the cables I have connected to the scope. This is bad, bad, bad. I carefully inspected the soldering I made to the crystal and to the PIC and saw no problems. It was time to play dentist so I pulled out my dentist's tool and scraped it carefully around the edges of the pcb-pad and track around pin 9 of the PIC and sure enough, there was a tiny bridge to small to see with the naked eye. I couldn't use the microscope because the PIC was in the way.
When you follow the pad around with this scraping tool and it suddenly stops or bumps (because there is something in the way) this means that there is solder or something

crossing the space between the pad and the ground plane or whatever is parallel to the pad or trace. I re-soldered the trim-capacitor as it was not the problem. I got a solder-braid and removed some of the solder on lead 9 of the PIC. I measured again and got about 7 MΩ of resistance between ground and pin 9. The bridge was removed. I connected the LCD and powered the unit up again and got the expected output, some random self-oscillation of the PLL. On my unit this was about 2.1 Gigahertz. I now soldered the last resistor, R5, the 100KΩ one. I connected power and LCD again and got 0.00 hertz oscillation. The Frequency Counter was ready and seemingly working alright!

Additional thoughts and ideas

The 0603 component pads on the pcb are large enough for 0805 components. I'm sure the counter would work fine with 0805 components as well. They are a little less hard to handle and solder. If you have an engraving tool you'd be better off making thermal clearance around the vias, on both the top and bottom planes.

The decoupling capacitor for the PIC should be placed closer to the PIC's VCC pin.

It is now time to test the unit by measuring some VCO's or other oscillators. I have yet to build a VCO in the Gigahertz range so I will get back to this. I have some Maxim samples lying around. I'll build something with those. A Bluetooth jammer would be fun to have around to annoy those annoying people with BT headsets.

Writing this document took approximately the same amount of time as it took building the unit. I hope someone will find it useful ☺

Disclaimer & copyright

Many thanks to Daniel Norman for a really cool kit. He sells some hard to find and very useful components at www.hem.passagen.se/communication/ especially components for RF designs.

Comments and corrections are welcomed, send me an email:
lasse@thelasse.com

created: 2006-10-16