INTRO | mki x es.edu

Hey there, thanks for buying this DIY kit! We – **Erica Synths** and **Moritz Klein** – have developed it with one specific goal in mind: teaching people with little to no prior experience how to design analog synthesizer circuits from scratch. So what you'll find in the box is not simply meant to be soldered together and then disappear in your rack.

Instead, we want to take you through the circuit design process step by step, explaining every choice we've made and how it impacts the finished module. For that, we strongly suggest you follow along on a **breadboard**[1]**,** which is a non-permanent circuit prototyping tool that allows you to experiment and play around with your components. To help you with this, we've included suggested breadboard layouts in select chapters.

In addition to this, you can also play around with most of the chapter's circuits in a **circuit simulator** called CircuitJS. CircuitJS runs in your browser. You'll find weblinks in the footnotes which will direct you to an instance that already has example circuits set up for you. We strongly encourage you to fiddle with the component values and general structure of those circuits to get a better understanding of the concepts we're laying out.

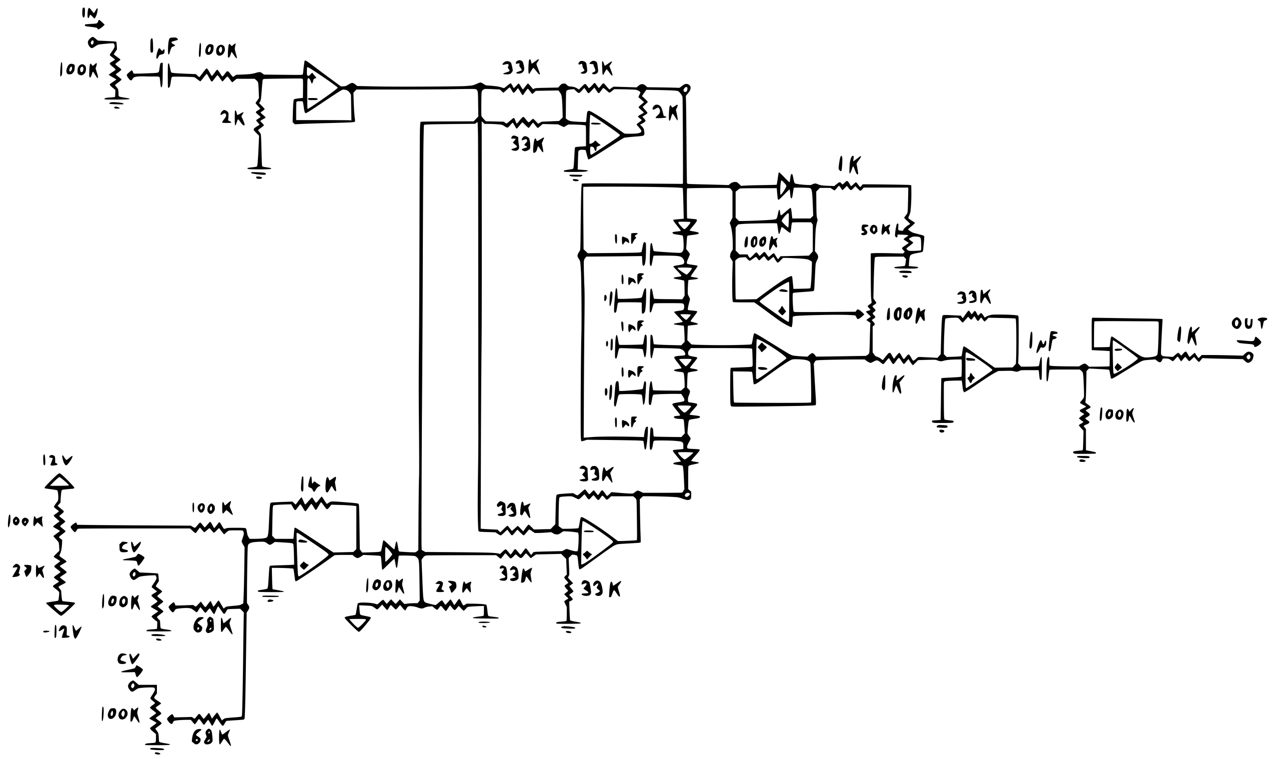
Generally, this manual is intended to be read and worked through front to back, but there were a few things we felt should go into a dedicated appendix. These are general vignettes on electronic components & concepts, tools, and the process of putting the module together once you're done experimenting. Don't hesitate to check in there whenever you think you're missing an important piece of information. Most importantly though: have fun!

TABLE OF CONTENTS

[1] Note that there is no breadboard included in this kit! You will also need a pack of jumper wires and two 9 V batteries with clips. These things are cheap & easy to find in your local electronics shop.

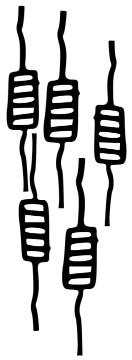
THE mki x es.edu VCF

Who doesn't love a good resonant low pass VCF? It's arguably the beating heart of any subtractive synth – adding character, dynamics, and a good deal of retro-futuristic oddness to your patches. That's why I set out to design a diode ladder VCF that sticks to super basic and widely available components. On top of that, I made sure that the architecture is as approachable as possible by avoiding the tricky diff-pair setup classic ladder filters rely on.



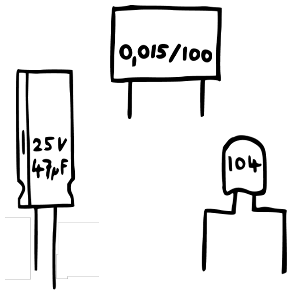
BILL OF MATERIALS

Before we start, please check if your kit contains all of the necessary components. In addition to a PCB, panel and power cable, your box should also contain:



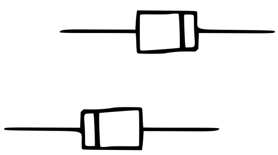
An array of resistors. The specific values (in ohms, which you should check for with a multimeter) are

100k	x5
68k	x2
33k	x8
27k	x2
14k	x1
2k	x2
1k	x3
10 Ω	x2



A bunch of capacitors. The specific values (which are printed onto their bodies) are

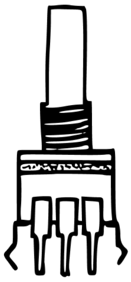
47 µF (electrolytic)	x2
1 µF (1J/film)	x2
100 nF (104/ceramic)	x6
1 nF (1n/film)	x5



Lots of diodes. The specific model names (which are printed onto their bodies) are

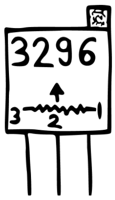
SB140[2] (schottky)	x2
1N4148 (signal)	x9

[2] Please note that these could also be a different model (e.g. 1N5818).



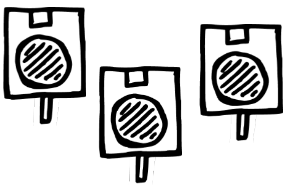
A couple regular potentiometers. The specific values (which may be encoded & printed onto their bodies) are

100k (B104) x5



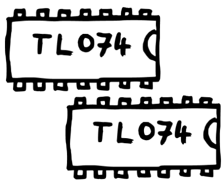
A trimmer potentiometer. The specific value (which is encoded & printed on top) is

50k (Y503) x1



A bunch of jack sockets. The specific models (which you can identify by their color) are

Switched mono (black) x4



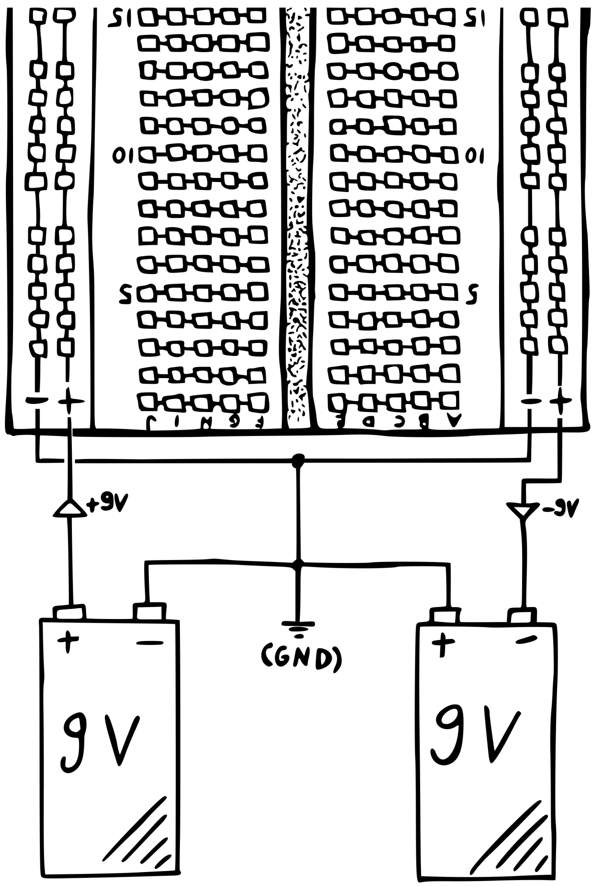
A couple of chips. The specific model (which is printed onto their bodies) is

TL074 (quad op amp) x2

You will also find a few sockets that are only relevant when assembling the module in the end.

POWERING YOUR BREADBOARD

Before we can start building, you'll need to find a way of providing your breadboard with power. Ideally, you'd use a dual 12 V power supply for this. Dual power supplies are great – and if you want to get serious about synth design, you should invest in one at some point. But what if you're just starting out, and you'd like to use batteries instead? Thankfully, that's totally doable. **You just need to connect two 9 V batteries to your breadboard like shown here.**[3] For this, you should use 9 V battery clips, which are cheap & widely available in every electronics shop.



By connecting the batteries like this, the row on the left side labeled **+** becomes your positive rail, the row on the right side labeled **+** becomes your negative rail, and both rows labeled **–** become your ground rails.[4]

Please make sure you disconnect the batteries from your breadboard when you make changes to the circuit! Otherwise you run the risk of damaging components.

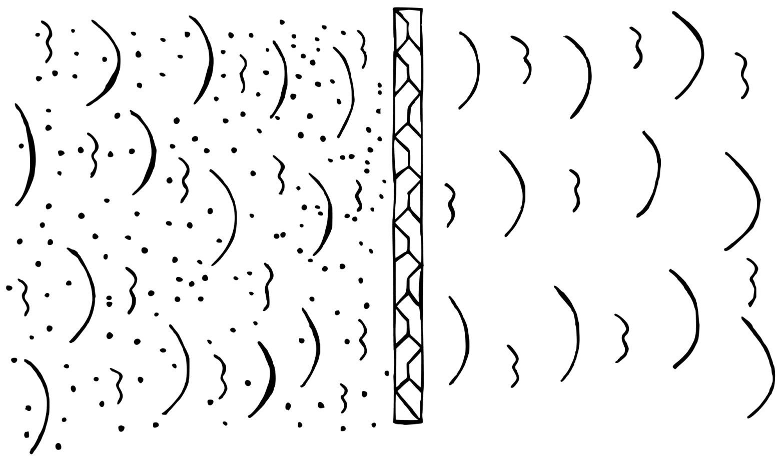
---

[3] Since all circuits in this manual were designed for a 12 V power supply, we assume that to be the default. Everything will still work roughly the same with 9 V, though.

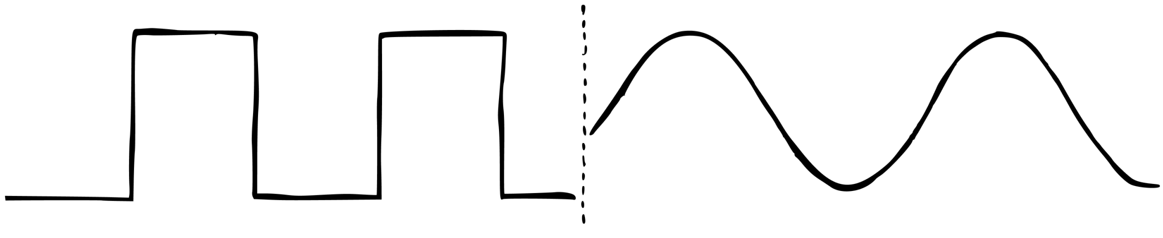
[4] This is a bit awkward because breadboards weren't really made with dual supply voltages in mind.

ANALOG FILTERING BASICS

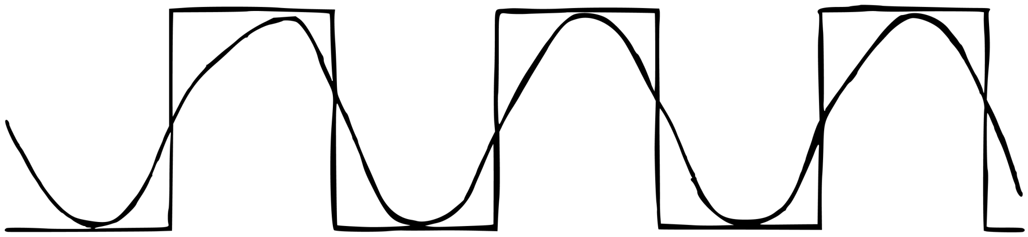
If you look at a bunch of schematics for voltage controlled filters floating around the internet, you'll quickly notice that they're quite convoluted and hard to figure out. Not only do they often use obscure and unusual components – they also arrange them in strange ways. This might lead you to believe that filtering is inherently tricky and complex. But that's actually not at all true!
If you strip away all the control-, amplification- and resonance circuitry, and just focus on the actual filtering that's going on, you'll find that most filters share the same, very simple core concept. To understand that core concept, we'll first have to talk about what filters do, exactly. **In this project, we'll focus specifically on low pass filters – which basically just take in an audio signal, remove a set amount of high frequencies and then send out a filtered signal that's mostly just bass and low mids**. In other words: they pass low frequencies.



Your flat's walls basically act just like that. If someone is talking in the next room, that person's voice will sound muffled and distant, because a lot of the high frequencies are filtered out. The music your neighbors are listening to will be filtered even more, since it has to pass through a bunch of additional walls.
Okay, but how does this translate to the world of electronics? It's actually really straightforward to visualize because we're able to look at different kinds of sounds with an oscilloscope. To make our lives a little easier, we'll check out sound-waves coming from an oscillator. These are way less complex and way more repetitive than most sound-waves you'll encounter in the real world. Which makes them easy to analyze.



On the left hand side, we have what's called a square wave. Square waves have lots of overtones and are therefore rich in high frequency content. In contrast, the sine wave on the right is tonally pure, which means that it has no overtones and no frequency content above its base pitch.
Let's now imagine that both are oscillating at the exact same frequency. This means that while their base pitch is identical, the square wave will contain all these high frequencies that make it sound really harsh, while the sine wave will sound mellow and rounded. **Coincidentally, if we send our square wave through a low pass filter, the result will actually look and sound a lot like a sine wave**. If that's the case – and you'll need to take my word for it for now – then we can make a rough assumption about which trait gives a waveform its high frequency content. Because if we overlay our square and our sine wave, we can clearly identify what sets them apart.

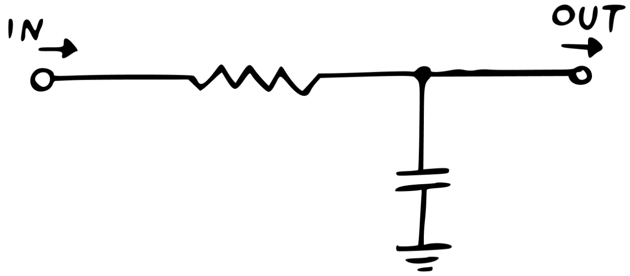


While the sine wave gently slopes upward and downward without any sudden change in direction, the square wave does exactly the opposite. It only deals in sharp, immediate 90° turns. And it's exactly those sharp turns that create a whole load of overtones. So as a rule of thumb, you could say that the pointier the waveform, the harsher it will sound. **And this leads us to a very basic definition of what filtering is in a visual sense**. **It's all about changing the shape of a given waveform**. Low pass filters sand down the edges of a waveform and round them out.
Now, you might ask how that definition helps us in any way – since we're basically talking geometry here and electrons don't actually take 90° turns. To answer that, we'll first have to think about what an electrical oscillation actually is. When we look at an oscilloscope and see a waveform – what's it drawing there, exactly? Well, in most, if not all cases, an oscilloscope will show you the voltage applied to its input terminals over time. Think of its display as a system of coordinates – with the x-axis showing time, while the y-axis is showing voltage. Our oscillator's sound-waves are then by definition recurring, characteristically shaped voltage swings.[5]
And that means that our square wave's sharp turns represent a very abrupt rise or fall in voltage, while the sine wave's soft slopes represent a way more gradual shift. **So in order to create a low pass filter, we will need to find a way to slow these abrupt voltage changes down**. Because remember: those pointy edges are what create all of the square wave's overtones.

[5] Read more about voltages, currents and resistances in the components & concepts appendix (page 43).

THE SIMPLEST PASSIVE LOW PASS

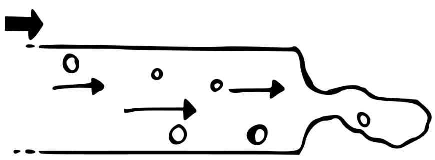
How do we do that? It's actually really simple. All we need are two components: a resistor and a capacitor.

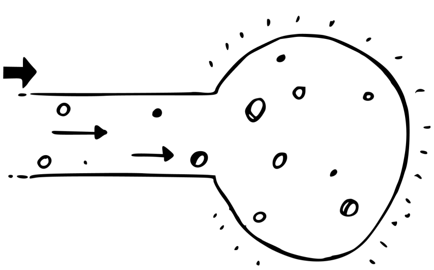


Set them up like this, and you get a basic low pass filter. Now any signal that we send in on the left will have some of its overtones removed. The result can then be picked up on the right. In case you're confused about how this works: let's dissect this little circuit. First of all, we'll take a look at what a resistor does in this scenario.

Resistors, if you don't know, basically act like narrow pipes that you can use to connect two points in your circuit.[6] Their resistance value, measured in ohms, determines how strongly they restrict the flow of current. **So by using a resistor to connect two points in our circuit, we are restricting the flow of electricity between them**.

This by itself, without the capacitor, would not achieve anything in regards to our waveform. As long as no current is actually flowing out of our output, no current is passing through the resistor. And this means that the voltage applied to the input just gets transmitted as-is. So the voltage on the right will always be exactly the same as the voltage on the left. Even if the resistor is really, really strong.



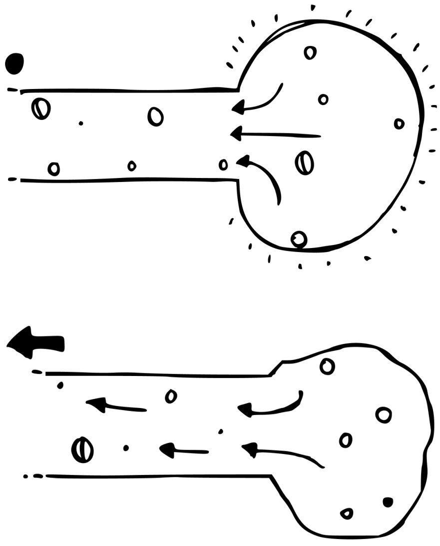


But as soon as we introduce that capacitor, our circuit suddenly behaves very differently. **And that's because capacitors basically act like balloons that you can attach to some point in your circuit**. These balloons come in different sizes, ranging from very tiny to very huge. We measure that in a unit called farad.

As these balloons are filled with electrical charge, they begin to expand.[7] **And just like with a real balloon, it will get harder and harder to push more charge in as the balloon starts pushing back increasingly**. Once the force we apply exactly matches the ballon's responding force, no current will flow anymore – the balloon is „full".

[6] Read more about resistors in the components & concepts appendix (page 45).

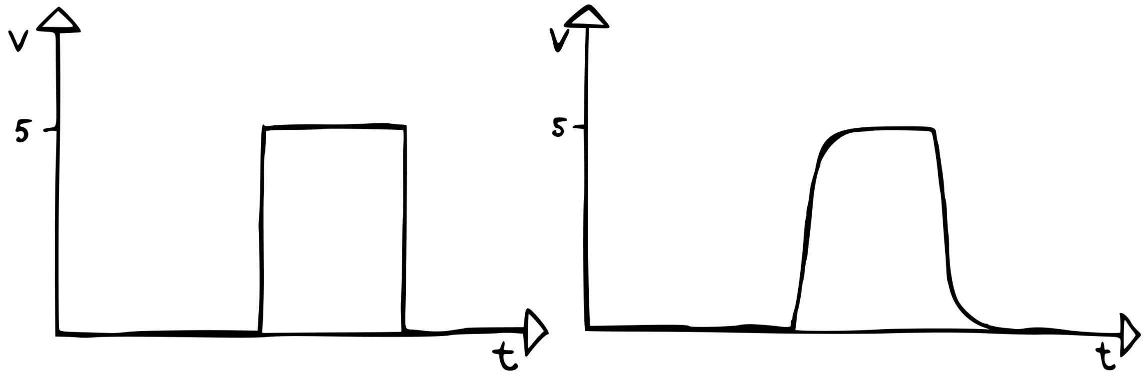
[7] Because the cap needs space to „expand into", we have to connect the second leg to ground.



But note that „full“ in this context simply means as full as the given pressure can force it to be. If we were now to increase the pressure, the balloon would fill up even more – until the forces are balanced again. Of course we can’t do this indefinitely. At some point, the balloon will pop – just like a capacitor can explode if you push it too far.

But let’s assume that we’re staying well within specifications. **Then once we stop pushing current into it, the filled balloon will push its contents back into the circuit.** As more and more current leaves, the pressure (i.e. voltage) within the balloon will drop. Until the two forces are once again balanced.

Going back to our filter circuit, let’s try and apply what we’ve just learned. We’ll keep track of the voltage levels at the input and output in these two graphs: input voltage on the left, output voltage on the right.



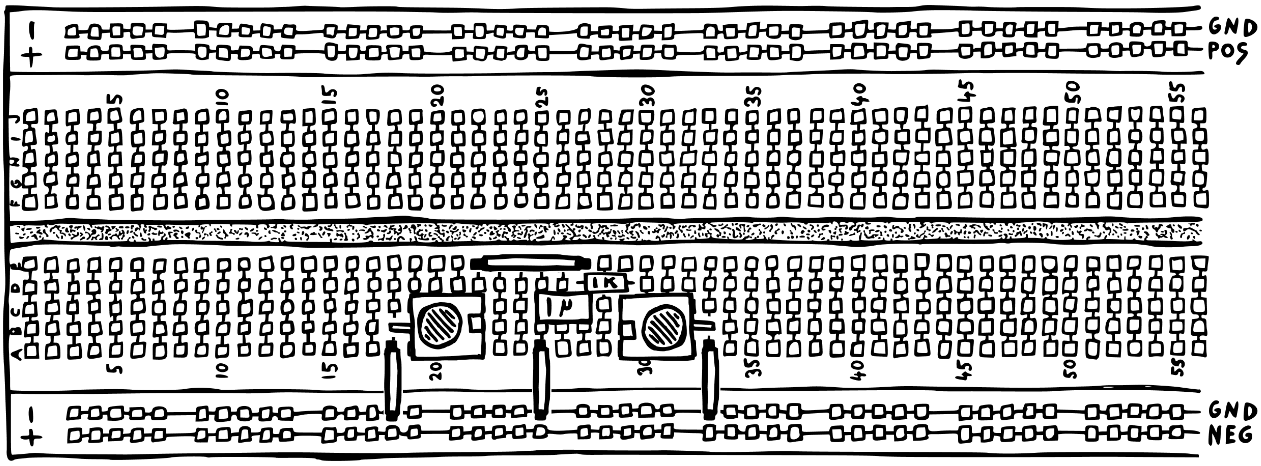
We now know that the resistor acts like a flow-restricting narrow pipe, while the capacitor acts like a charge-storing balloon. So let’s say that the input voltage suddenly jumps from 0 V to 5 V. Unlike before, that voltage will now cause current to flow through the resistor and into the capacitor, filling it up.

This means that the input voltage is no longer instantly transmitted to the output – because filling the capacitor up takes some time. Once it is full, we’ll read 5 V at the output. And the rate at which it fills up depends on two key factors: the size of the capacitor and the strength of the resistor. The bigger the capacitor, the longer it takes to fill it up. The weaker the resistor, the more current can flow and the quicker the capacitor is filled up.

Next, we’ll imagine that the input voltage suddenly drops to 0 V. This means that the capacitor will start pushing its contents back through the resistor, since there is no opposing force anymore. **While the capacitor is emptying, the output voltage is slowly declining – until it finally hits 0 V**. And again, the duration of that process depends on the capacitor’s size and the resistor’s strength.

You'll notice that I've drawn a slight quirk into the way the output voltage ramps up and declines. **That's because pushing current into the capacitor gets harder as we approach the maximum voltage level, and the voltage inside the capacitor gets weaker as it empties out**.[8]

The end result is a waveform that is decidedly less angular than the square wave we sent in, even though it's still quite far away from being a pure sine wave. But we can nevertheless expect it to sound a lot less harsh. To try this, get your breadboard and dig out a 1k resistor and a 1 µF capacitor from your kit.



If you now connect a square wave signal to the circuit's input, you should be able to pick up a much more mellow version at the output. Cool – but what if we want to adjust the amount of filtering we apply to our square wave? It's actually quite simple. The longer it takes for our capacitor to charge and discharge, the more overtones will be removed. **In synth terms, we often talk about this as the filter's cutoff point**. This describes the frequency at which a filter will begin filtering.

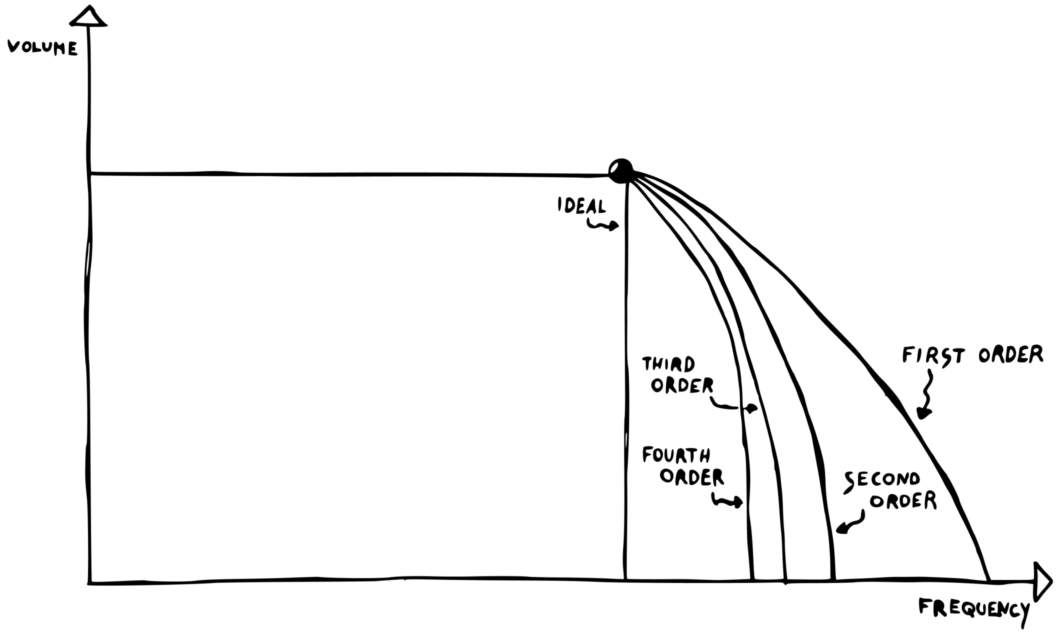
For our low pass filter, that means we have two options if we want to move its cutoff point. We can change the value of either the capacitor or the resistor. So find some different resistors and capacitors in your kit and try them out in the circuit. You can even replace the resistor with a potentiometer (set up as a variable resistor) to change the cutoff frequency on the fly.[9] **One caveat, though: don't try the electrolytic capacitors here.** These are polarized, which makes them unfit for use with audio signals in this scenario.

[8] You can try this chapter's circuits in a circuit simulator. I've already set them up for you right here: https://tinyurl.com/y5a8tc9l – you can change all values by double clicking on components.

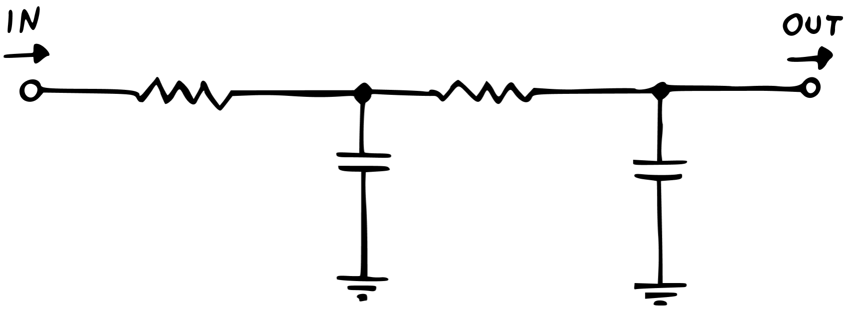
[9] Read more about potentiometers in the components & concepts appendix (page 49).

THE MULTI-STAGE LOW PASS

Now, you might have noticed that our filter does a pretty bad job at eliminating the high-end effectively. You could probably hear a lot of unwanted frequencies bleed through. This is because it is a simple first order filter, which means that there's just one single filtering stage.



An ideal filter would be able to completely eliminate all frequencies beyond its set cutoff point. A first order filter, on the other hand, does really not live up to that standard. Instead, it gradually reduces the volume after the cutoff point – by 6dB per octave, to be precise. In many applications, this is just too weak a performance. So what can we do to give our filter more bite?

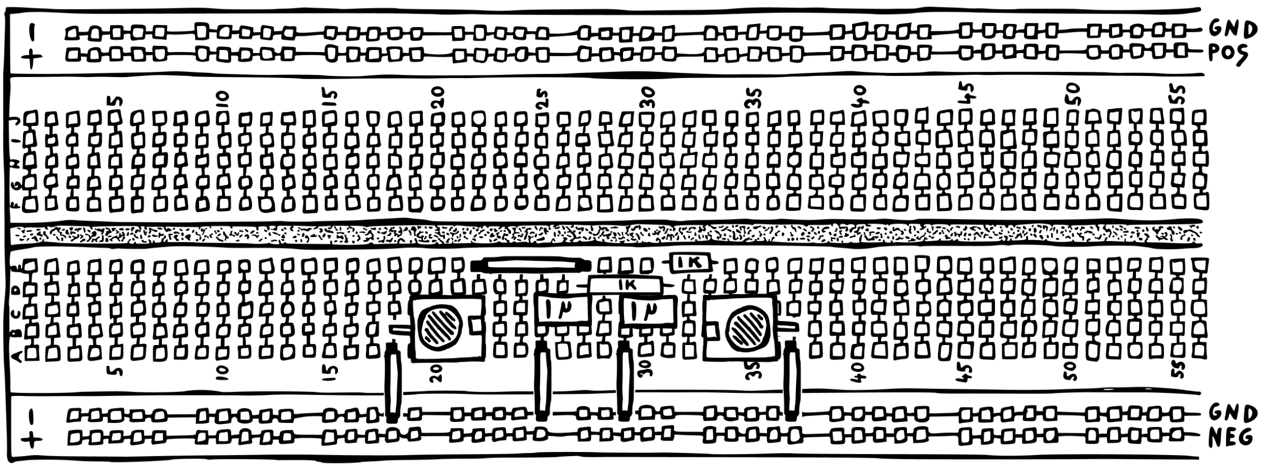


It's quite simple: we just chain multiple filtering stages together. Two will make a second order filter. Three a third order. And four a fourth order – and so on. For every filtering stage we add, we will get a steeper roll-off. **And that kinda makes sense: because we're basically just applying the same filtering process to an already filtered signal**.

Think of it as if you're screening sand – and then you're screening the screened sand again.[10]

There's just one catch if we want to try this out on the breadboard. **For a second order filter to work properly, the resistors and capacitors used in both stages need to be of the same value – so that both stages operate with the same cutoff frequency**. Now, if we use regular resistors, that isn't really a problem. But in case you want to adjust the cutoff frequency on the fly again, you'll be running into a bit of an issue here.

You could of course use two same-value potentiometers and then try to adjust them in lockstep. But since this is impractical and imprecise, that wouldn't really be fun – so I suggest you stick to regular resistors for this one.

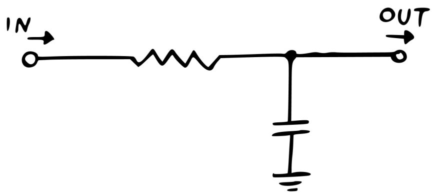


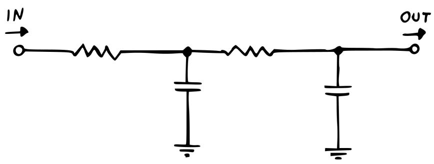
If you'd now check out the output signal on an oscilloscope, you'd see that unlike before, the result is much closer to being a sine wave – and so the high-end is nearly inaudible.

[10] You can try this chapter's circuit in a circuit simulator. I've already set it up for you right here: https://tinyurl.com/y5x873gn – you can change all values by double clicking on components.

THE ACTIVE LOW PASS

As a next step, we'll turn our design from a passive into an active filter. Why is that necessary? Well, first of all, I wasn't completely honest about the performance of our circuit. **While it's tempting to think that simply chaining two passive filters gives you a true 2-pole response, this is sadly not the case.** To understand why, we'll have to go back and re-examine how those individual stages work.

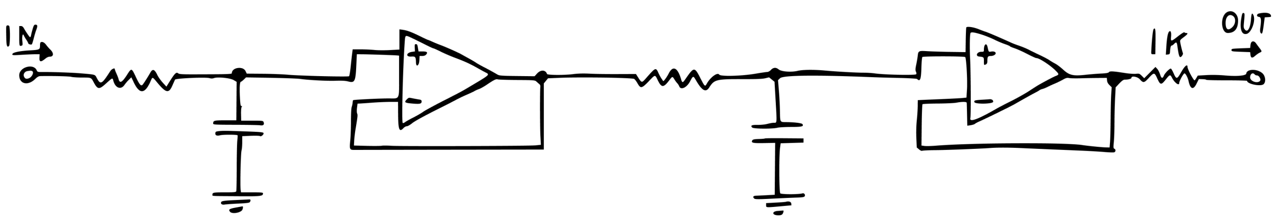




As we just learned, the filtering process is basically just an interplay between a resistor and a capacitor. We push a current through the resistor and into the capacitor. As the capacitor fills up, the voltage above it slowly rises. Now, as long as that interaction between resistor and capacitor is undisturbed, everything goes according to plan.

But if we chain two filter stages, the story changes quite a bit. **Because where before, the first stage's output was completely sealed shut (as long as we don't plug anything into the output), now there's an opening there.** Meaning that an additional path opens up for our current while the capacitor is filling up. This, of course, is messing with the way the first filter stage operates. But it also affects the second stage, since that second stage only gets the scraps of the first – the current that spills over.

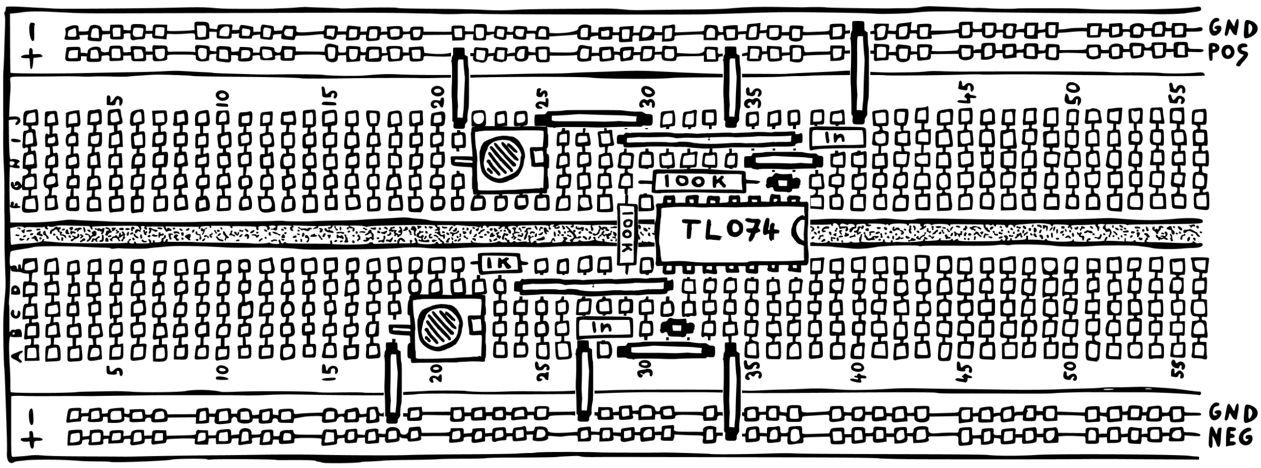
Because of this interdependence, the filter's performance suffers as a result. Instead of a real 2-pole response, we get something much less cleanly defined. How do we fix this? You guessed it – by adding amplification to our design and turning it into an active filter. To do that, we have a variety of options – but for simplicity's sake, we'll go for the classic solution: using operational amplifiers – or op amps for short.[11]



If we set two op amps up like this, we can use them to buffer the voltages at both our filter stages' outputs. **Buffering, in this context, means that we provide a perfect copy of a voltage level without interfering with that voltage level in the process.** This way,

[11] Read more about op amps and buffers in the components & concepts appendix (page 51).

both stages are allowed to work in complete isolation – without any interdependence or disturbance by outside influences. Improving our filter's performance noticeably.[12] Finally, we put a 1k resistor between the second stage's buffer and the output socket to protect the buffer from short circuits.
To try this out, we'll be using a chip called the TL074. It contains 4 discrete op amps – and although this might seem like overkill for now, we'll be needing all of them later on. **Make sure to set it up exactly as shown here – if you reverse the power connections, it will heat up and die!**

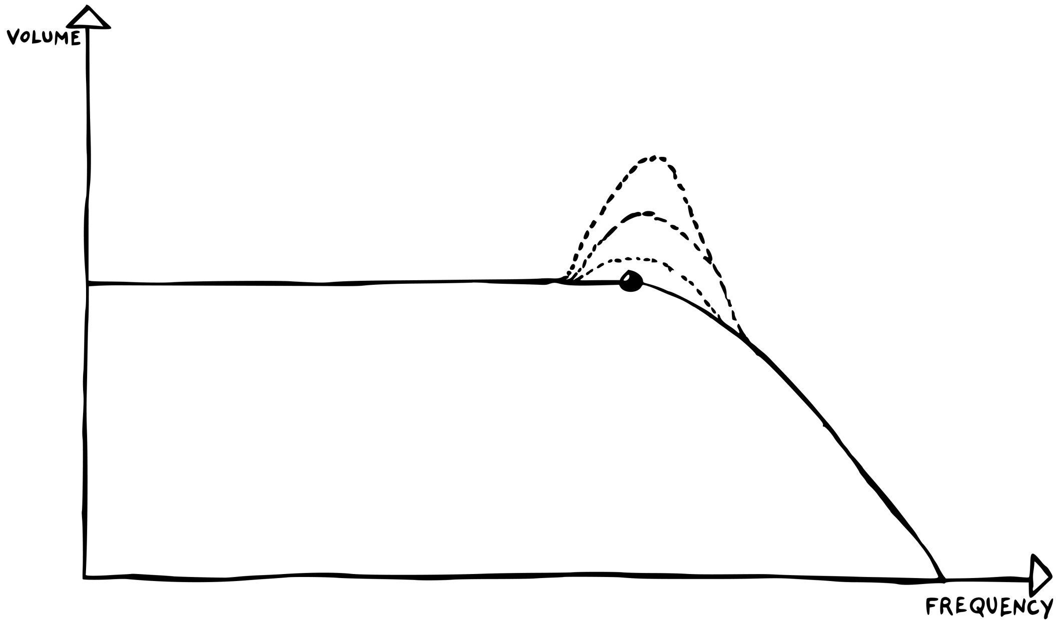


And that's it – we have an active filter! Patch a square wave into the input and see how it performs.

[12] You can try this chapter's circuit in a circuit simulator. I've already set it up for you right here: https://tinyurl.com/y2fcoqsj – you can change all values by double clicking on components.

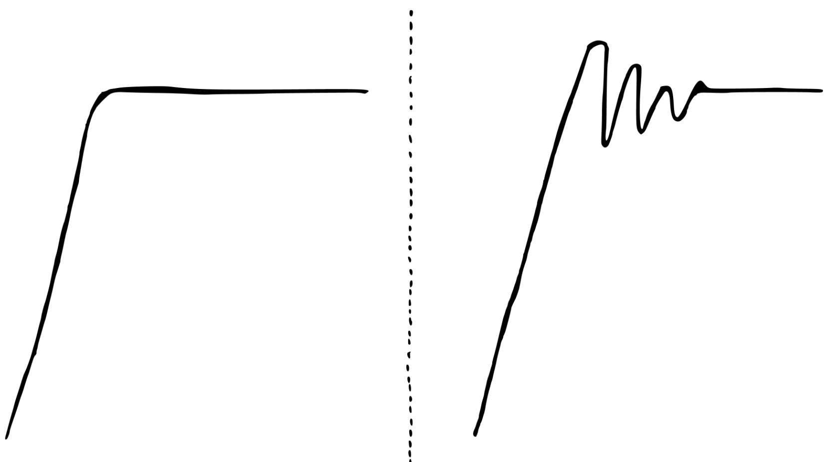
RESONANCE BASICS

So far, so good. But a filter is not quite complete without resonance. It's what gives it that futuristic sound, after all. Think of a typical squelchy acid bass-line: the filter seems to be adding something to the sound there.



And here's how that's normally visualized. As long as the resonance is completely turned down, the filter will simply remove the frequencies above the cutoff point without doing anything else. But as we turn the resonance up, the filter will also start to amplify the frequencies around that cutoff point. **So instead of just removing frequencies, the filter starts to also emphasize some.**

Unfortunately, this type of visualization doesn't really help us figure out how to make our filter resonant. Yes, we know that we have to amplify those frequencies around the cutoff point – but how do you do that? To get a better idea of what resonance added to a waveform actually looks like, let's try a different type of visualization.

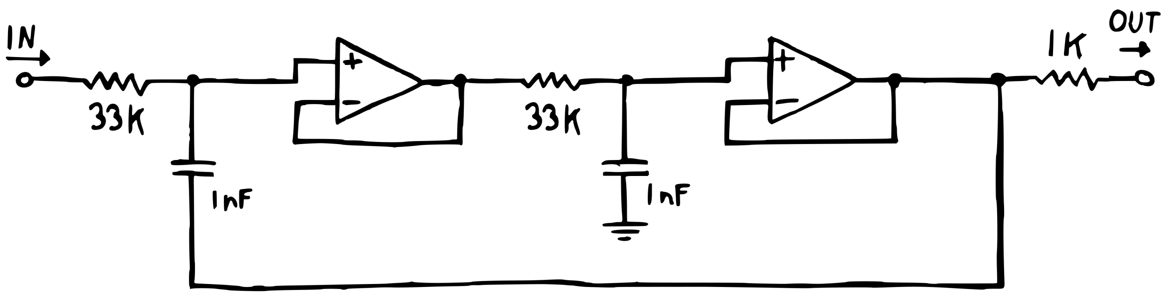


On the left, we've got a lightly filtered square wave without resonance. And on the right, we've got the same input but with a good amount of resonance dialed in. As you can see, something weird's going on with the rising edge. It looks like as if the wave has trouble stabilizing at the peak voltage level.

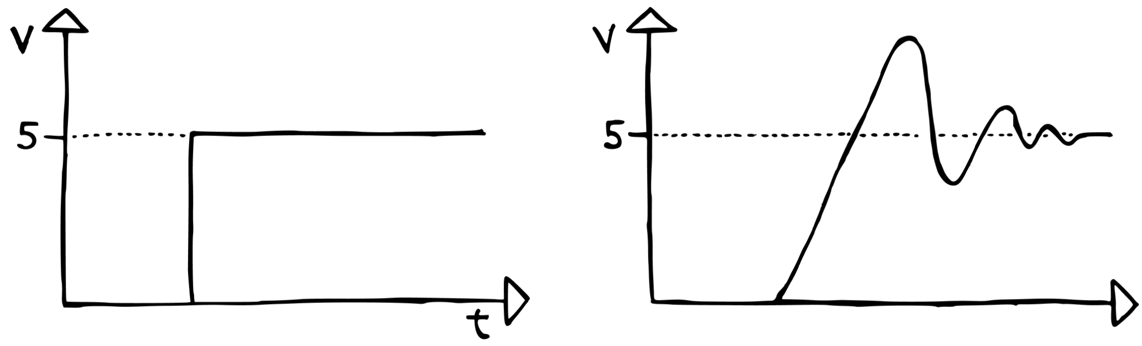
Where on the left, we have a nice and smoothly rounded edge, on the right, we get this over- and undershooting behavior that only settles down over time. And if you'd turn up the resonance further, this behavior would only get more intense. Also, if you'd lower the cutoff point, the wave would stay in the individual over- and undershooting phases for longer and longer.

This gives us a rough idea of what separates a non-resonant from a resonant filter. While the former does its job (namely filtering) in a precise and controlled manner, the latter really struggles with that task. **It seems as if the resonant filter overreacts to any sudden change in voltage at its input, leading to this sloppy, bouncy behavior**. But that bouncy behavior is not random. The time spent in the over- and undershooting phases is constant as long as the cutoff point stays in place. So it's safe to assume that the frequency of this added oscillation is directly tied to the cutoff point.

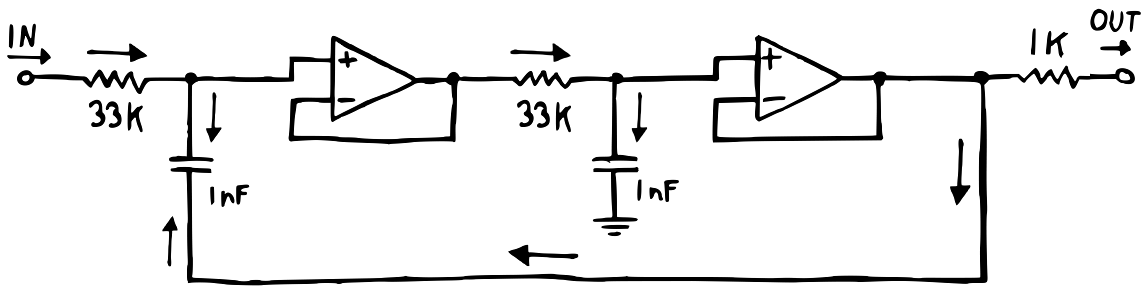
If that's the case, the notion of the raised cutoff point starts to make sense: **resonance, then, is basically just the filter itself swinging at its set cutoff frequency in addition and reaction to the oscillation it's supposed to be filtering**. As our oscillator swings, the filter swings along at its own pace. And while this might seem very tricky to implement, in its most basic form, it's actually really simple.



All we have to do is route the second filtering stage's output to the normally ground-connected leg of the first stage's capacitor. In case you're confused about what this does: let's analyze this circuit step-by-step.

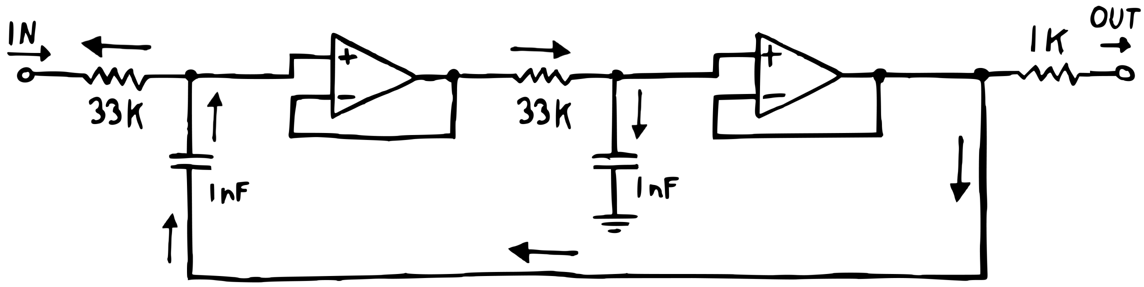


We’ll once again keep track of the in- and output voltage levels in two separate graphs. Like before, we’re assuming the input signal to be a square wave. Then, we can expect the input voltage to rise to 5 V very abruptly, causing a current to flow through the first resistor and into the first capacitor. This is possible because the filter’s output rests at 0 V in the very beginning.
The thing about op amps is that they are not only able to push out current – they can sink current as well. **So connecting the capacitor to the output is virtually the same as connecting it straight to ground at this moment**. It’ll basically „expand“ into the op amp’s output, allowing it to be filled up the same way as before.
But as the capacitor fills up, the adjacent buffer will start raising the voltage at its output, forcing a current through the second resistor and into the second capacitor. This, in turn, will cause the next buffer to slowly raise the voltage at its output. **And since that output is connected to the first capacitor, something odd happens: a voltage starts pushing up from below the capacitor**.

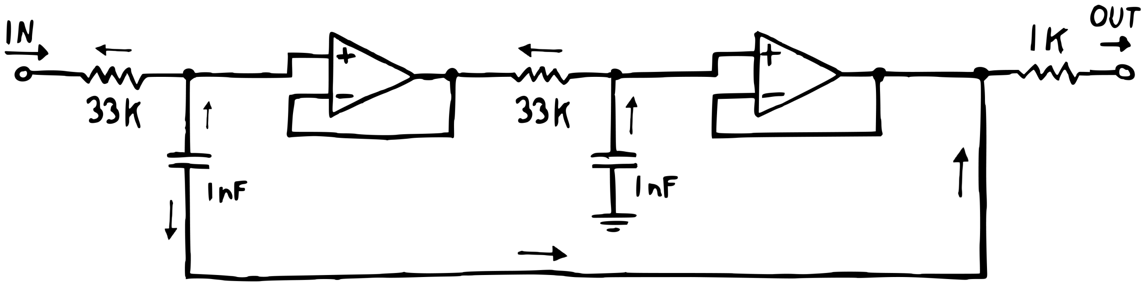


Now, as long as that voltage is still relatively low, current will keep flowing through the first resistor and into the capacitor – though at a slower and slower rate because of the pushback. As that pushback increases, the voltage above the first capacitor will be raised to 5 V. Under normal circumstances, this would be the peak voltage level. **But since it takes some time until the filter’s output has reacted to whatever happens in the first stage, it will keep raising the voltage**.
As that voltage pushes harder and harder up against the first capacitor, the voltage above it will be raised beyond 5 V. We are effectively creating an „overpressure“**.** Which, in turn, will see an additional, delayed rise in voltage at the filter’s output – pushing the voltage in the first stage even higher. This cascading effect will drive the output way beyond 5 V –

and it would keep doing that indefinitely, were it not for the first resistor. **Because since the input voltage is now lower than the voltage within the first stage, current will start flowing out of the capacitor and through the resistor – into the input**.



As the capacitor now fills up from below, the voltage above it will slowly begin to fall due to the resistor acting like an equalizing valve. This kicks off a cascading effect in the opposite direction from before – because a drop in voltage within the first stage will cause a delayed drop in voltage at the filter's output. Which in turn takes some pressure off of the capacitor, lowering the voltage above it even further.

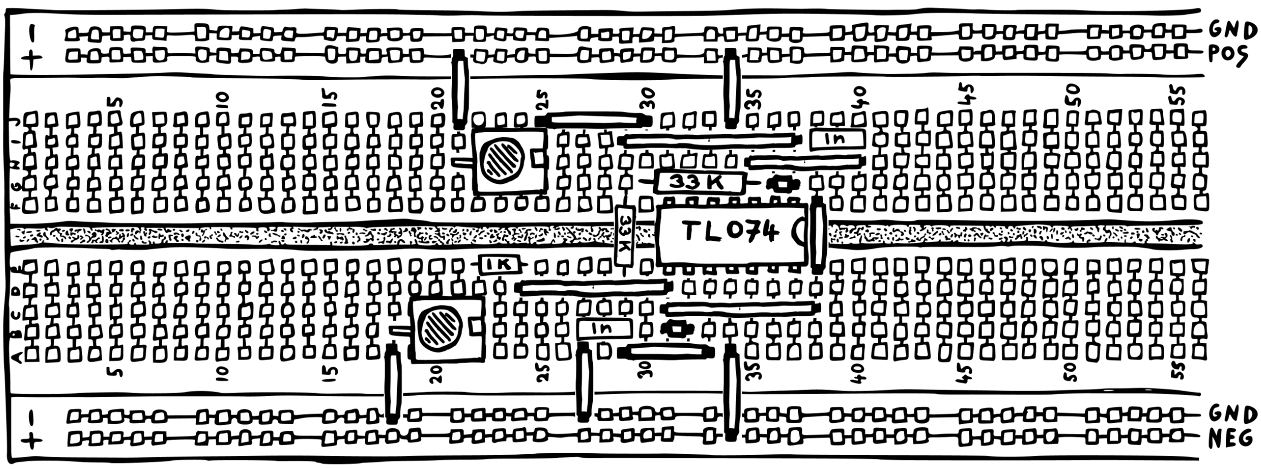


Eventually, we'll even drop a fair bit below 5 V – which means that the current going through the first resistor reverses its direction and the whole process repeats. But since this time, we're not starting with a completely empty capacitor, the cascading effect has decidedly less time to ramp up – and as a result won't push the output voltage as high as before.

After a while, the over- and undershooting will slowly fizzle out as the output voltage stabilizes at 5 V. And that's it: we've made our filter resonant.[13] By now, it should've also become clear why that resonance causes the filter to swing at its set cutoff point: **the time spent in the over- and undershooting phases is directly determined by the capacitor sizes and resistance values**. Which, in turn, also determine the cutoff point. But enough theory: let's see if this actually works.

---

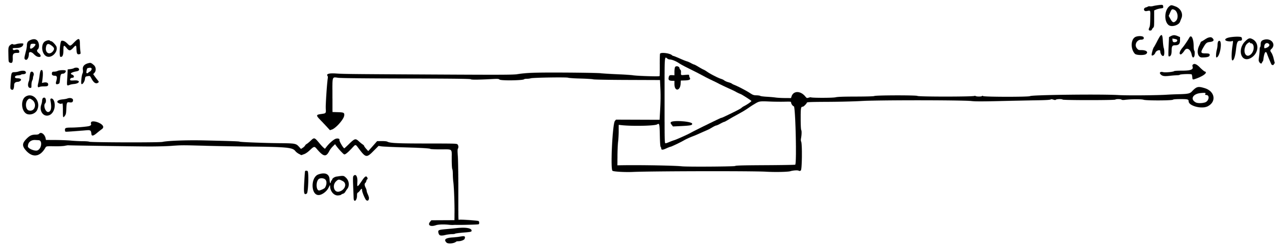
13 You can try this chapter's circuit in a circuit simulator. I've already set it up for you right here: https://tinyurl.com/y4j2khjd – you can change all values by double clicking on components.



If you plug a square wave signal into the input again and listen to the output, you might be a bit disappointed: there's not really an audible difference. So what's going on here? Simple: we're not applying enough feedback to the first capacitor for it to have a noticeable effect. We said earlier that an increase in resonance corresponds to a more intense over- and undershooting behavior. **So it's safe to assume that increasing the amount of resonance can be done by increasing the amount of feedback we apply to the capacitor.**

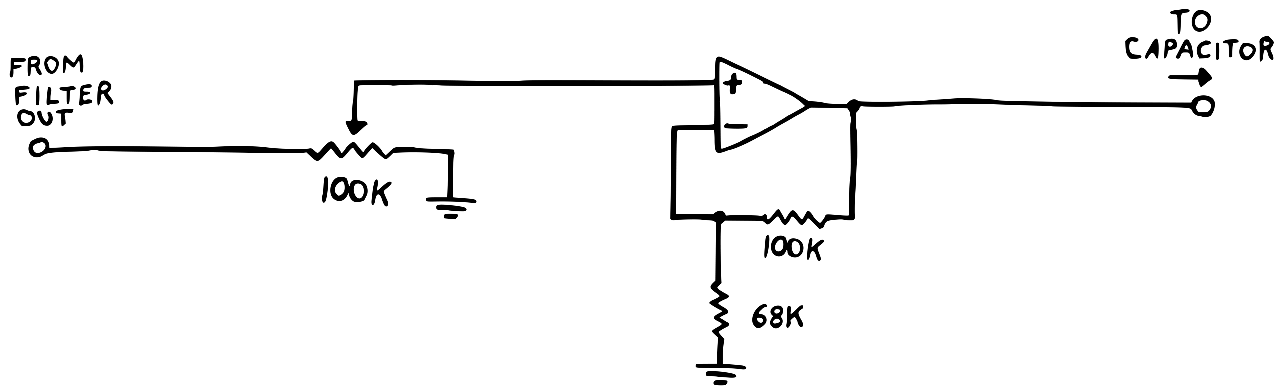
VARIABLE RESONANCE

But before we go about increasing the amount of of feedback, let's first talk decreasing it, since that is a bit simpler to pull off. All we really need to do is scale the filter's output down before sending it to the capacitor. How do we do that? Easy – we use a potentiometer that we set up as a variable voltage divider.[14] This way, we can adjust the feedback volume by turning a knob.



But before we send the result to our capacitor, we need to route it through another buffer. **That's because a voltage divider is limited in the amount of current it can source and sink – since that current will have to pass through a significant amount of resistance**. With a buffer, our capacitor can get filled up and drained as needed without any impediments.

Okay, so with this, we'd now be able to decrease the amount of resonance. How about increasing it though? To do that, we'll have to turn our buffer into a proper amplifier by replacing the straight connection between inverting input and output with a voltage divider. **This forces the op amp to work harder and thereby increases its gain.**[15] The proper amount of gain for the amplifier is something that should be tweaked to taste though. For now, we'll be using a 100k/68k voltage divider.

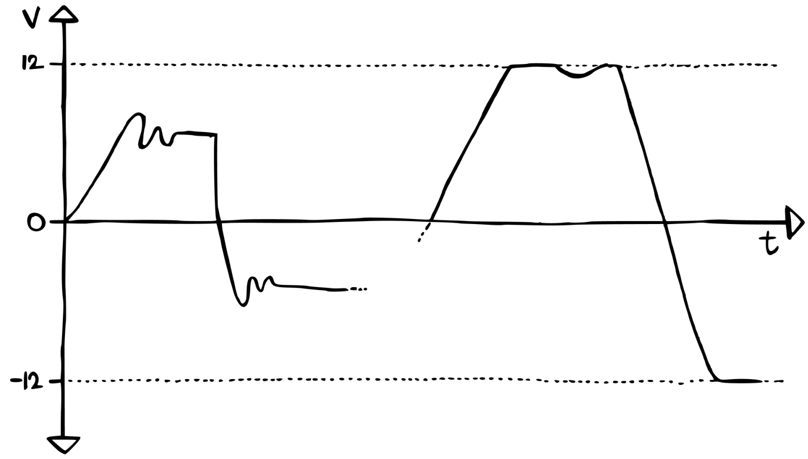


If you'd set this up on your breadboard and proceeded to test it by sending in our square wave again, you'd run into an unfortunate problem. As you'd turn up the resonance, the

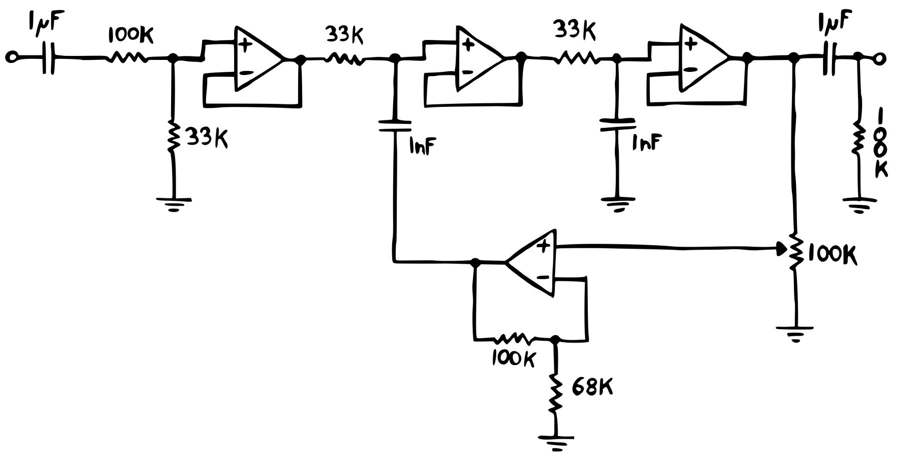
[14] Read more about (variable) voltage dividers in the components & concepts appendix (page 48/49).

[15] Read more about op amp amplifiers in the components & concepts appendix (page 52).

sound would get distorted more and more – and in my opinion not in a good way. Why is that?
Well – it's all about headroom. Since we're supplying our op amp with + and -12 V, the range in which it's able to operate is strictly defined. This means that it can only accurately amplify signals if it stays within those boundaries. **If we push it too far, the feedback amplifier will basically get stuck at the minimum or maximum voltage level that it can put out.**

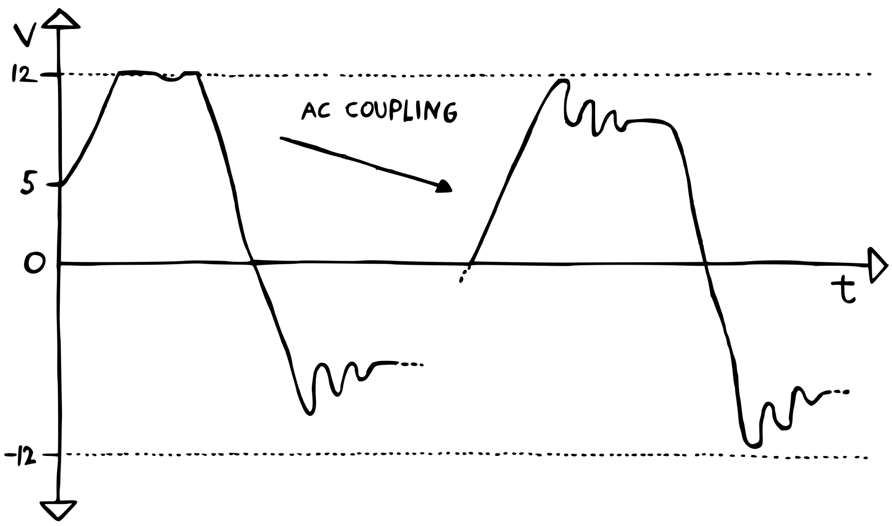


So what happens is that the cascading effect is ramping up so quickly and intensely that the op amp runs into its limits almost instantly. To fix it, we we have two options. We could simply lower the feedback amplifiers's gain. And this would eliminate the problem – but it would also limit the maximum amount of resonance we can add to the sound. **The second option is much more viable: reducing the filter's input volume**. Because if the input volume is lower, we'd slow down the cascading effect, while also leaving more overall headroom for the resonance. How do we do that?

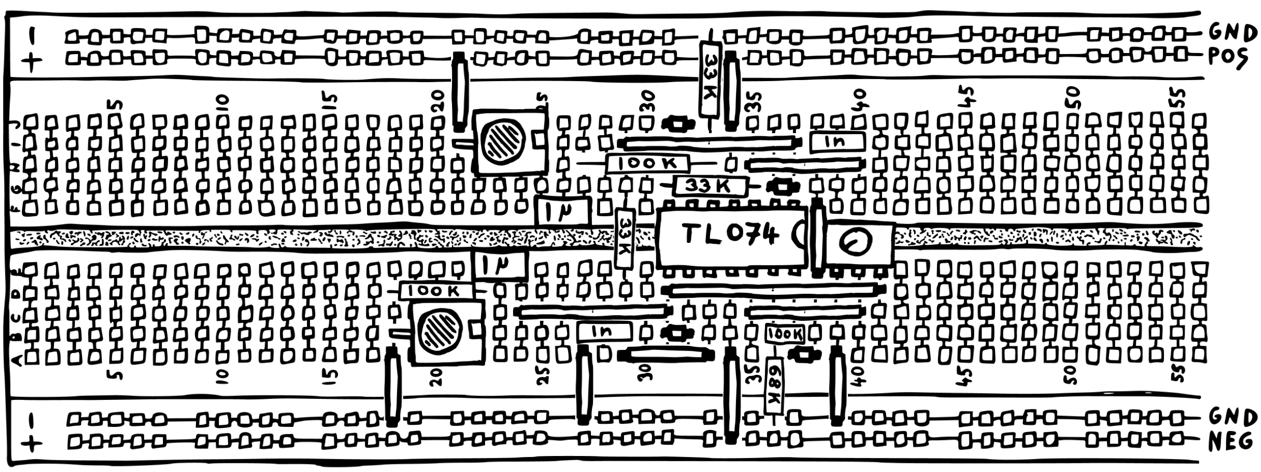


We already know the solution: by combining a voltage divider with an op amp buffer. You could of course use another potentiometer as a variable voltage divider if you'd like to experiment with the volume balance. But in order to keep it simple for now, we'll just use a fixed 100k/33k divider.

Also, we'll add AC coupling before the input and after output.[16] **This way, we're making sure that both the in- and output signals are properly centered around the 0 V-line**. Additionally, the capacitors will act as a very effective short circuit protection, since they block constant current flows. That's why we can safely leave out the 1k resistor after the output. And at the input, we can save us an additional 100k resistor going to ground after the capacitor, simply because the 100k/33k voltage divider is already providing a resistive path to ground.
Okay, but why do our in- and output signals need to be centered around the 0 V-line? Simple: because if they weren't, we might run into more headroom issues.



Imagine our input wave is swinging off-center, with its center shifted upwards by 5 V. Then our feedback amplifier will quickly crash into its upper voltage limit, causing the resonance to sound distorted and horrible again. **To avoid issues like that, it's good practice to add AC coupling to every audio processing module's in- and outputs**.[17]



[16] Read more about AC coupling in the components & concepts appendix (page 22).

[17] You can also try this chapter's circuit in a circuit simulator. I've already set it up for you right here: https://tinyurl.com/2bfrjd36 – you can change all values by double clicking on components.

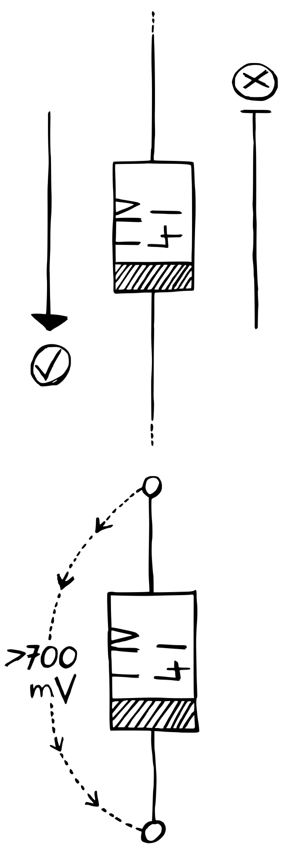
If you test this out now, you should be able to hear that the resonance does not cause our filter to distort the signal. **Instead, we're able to drive it into self-oscillation territory**. This is when the cascading effect gets so strong that it just never settles down. It will even keep going if you disconnect the input. This way, you can even use the filter as a fairly decent sine wave oscillator!

DIODES AS RESISTORS?

So far, so good. But our current design has one serious flaw: it doesn't allow you to change the cutoff frequency. Or at least not in a convenient manner. Ideally, you'd want your filter to expose a CV (control voltage) input, so that you can control its cutoff frequency using a voltage. Implementing this is not trivial, though. On the contrary: it's probably the most challenging aspect of analog filter design.

To understand why, we have to briefly recap what determines the cutoff frequency in our current design. We said that the filtering process is essentially an interplay between a resistor and a capacitor. The capacitor gets filled and drained through the resistor. The longer that takes, the more overtones will be removed, and the lower the cutoff frequency will be. To adjust the cutoff frequency, we can speed up or slow down that process by varying either component's value. **So to implement voltage control over the cutoff frequency, we'd either need a capacitor or a resistor that changes its value in response to an applied voltage**. Unfortunately, neither of those things exist in a neat, reliable package.

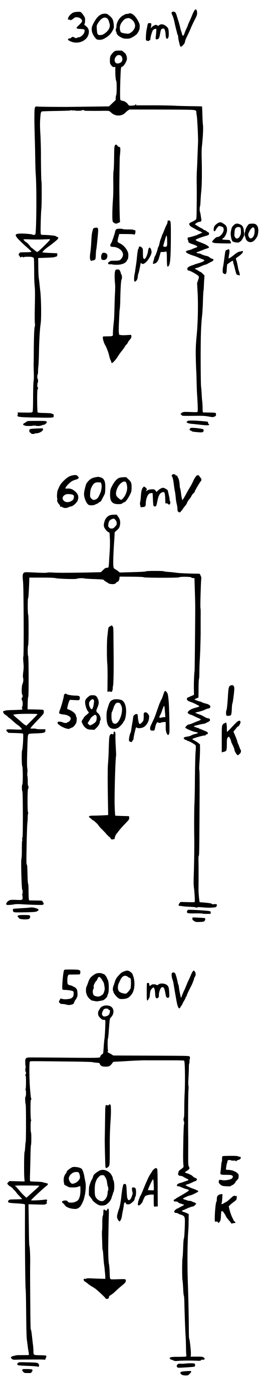
This forces us to explore less-than-intuitive alternatives – one of which is using diodes. How does this work? Surely you can't just use them to replace the resistors in our original filter design? Actually, you kinda can! Not without some caveats though, of course, which we'll get to in a minute. But generally, a diode can indeed be used as a quite capable voltage controlled resistor.



Before we get ahead of ourselves though, let's take a step back and recap the absolute basics here. What are diodes, exactly? If you look that up online, chances are the first thing you'll read is that a diode is „an electrical component that allows the flow of current in only one direction" – so essentially a one-way street for electricity. And while this is of course true, it's also an oversimplification. There is one major caveat that you'll probably read about right after that main definition: **diodes only start conducting if the voltage at their input terminal is 0.7 V larger than the voltage at their output terminal**. So the input signal needs to be strong enough to „activate" the diode.

Often, this is modeled in a kind of digital way: as if the diode had clearly defined on- and off-states. We could also think of those as two resistive states: no resistance, infinite resistance. But of course, this is again way oversimplified. In reality, a diode won't just abruptly turn on as you increase the input voltage from 699 to 700 mV.

That's because a diode is not a digital device – and therefore does not have those clearly defined on- and off-states. Instead, it's messy and analog. **This means that a diode will never be completely closed – but rather open to varying degrees, depending on the input voltage**.

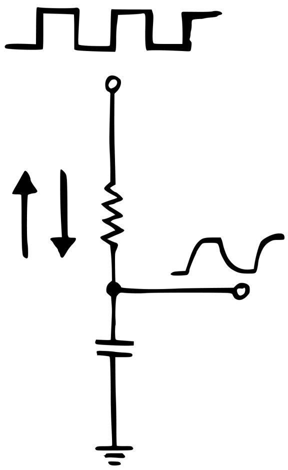


Think of it like this: let's say we take a diode and apply 300 mV to the input, while connecting the output to ground. Now, if we try measuring the current flowing through the diode, we'll see two things. First – that there actually *is* a current flowing. And second, that that current is quite small – 1.5 µA in my experiments with a generic 1N4148 diode.

This means that it is not blocking the current – it's just strongly resisting it. So if we replaced the diode with an adequately-sized resistor (around 200k), our circuit would behave virtually the same: we'd still have 1.5 µA flowing from top to bottom. **Which makes it safe to assume that under these specific circumstances, our diode will behave just like a 200k resistor**.

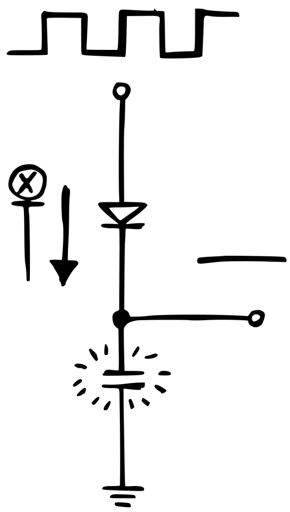
Note that the emphasis here is on „these specific circumstances". Because as soon as we in- or decrease the input voltage, the comparison falls apart completely. For the resistor, a doubling in voltage will result in a doubling in current – so at 600 mV, there'd be 3 µA flowing. But with our diode, the increase in current will actually be exponential. So at 600 mV input voltage, there'll be 580 µA going through our circuit. That's 386 times more current – and so the comparison to the 200k resistor doesn't work at all anymore. Instead, the diode now behaves more like a 1k resistor. And if we somewhat decrease the input voltage to, say, 500 mV, a 5k resistor would be the closest match. **So essentially, you could think of the diode as a voltage controlled resistor – where the voltage applied to the input terminal directly determines the diode's resistance value.** Which is exactly what we're looking for! Except there's one major problem.

How do we get our oscillation through the diode, then? Because by definition, an oscillation is a changing voltage over time, and if we change the diode's input voltage, its effective resistance will change wildly in response! Worse yet – since diodes are one-way-streets for current, they will cut off the lower half of a regular audio signal. Because remember, those are centered around 0 V – and if the input voltage goes negative, the diode will block.



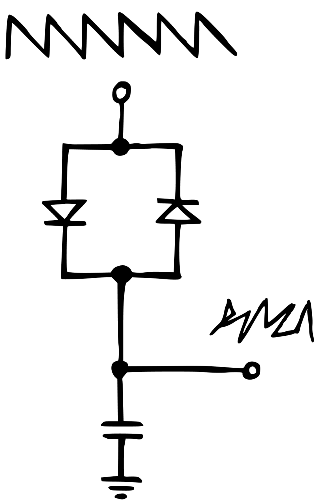
To really gauge the problem, let's re-examine a single passive low-pass filter stage. The basic working principle there was this. We fed an oscillation into the resistor, which would cause a relatively small current to flow into and out of the capacitor, filling and draining it slowly and thereby rounding the edges of the wave.

This is possible because the resistor doesn't care about the current flow's direction. It's going to behave the same either way. Also, the voltage we apply will have no effect on the resistor's resistance value – so we can vary it as much as we want without affecting the filtering process. Now, if we were to simply replace the resistor with a diode, there'd instantly be a rather glaring issue.



The current return-path would be blocked, making it impossible for the capacitor to discharge. As a result, it would just get filled up to the brim and then stay that way forever, effectively turning our oscillation into a steady high voltage. Which is rather unfortunate. So we'd need to find an alternate route for our capacitor to be drained through.

Now you might be inclined to ask: why not just use another diode that leads back to the source signal? This way, the high phase of our wave could push through one, while the low phase could drain the cap through the other diode. Problem solved, right? Yes, but only partially.

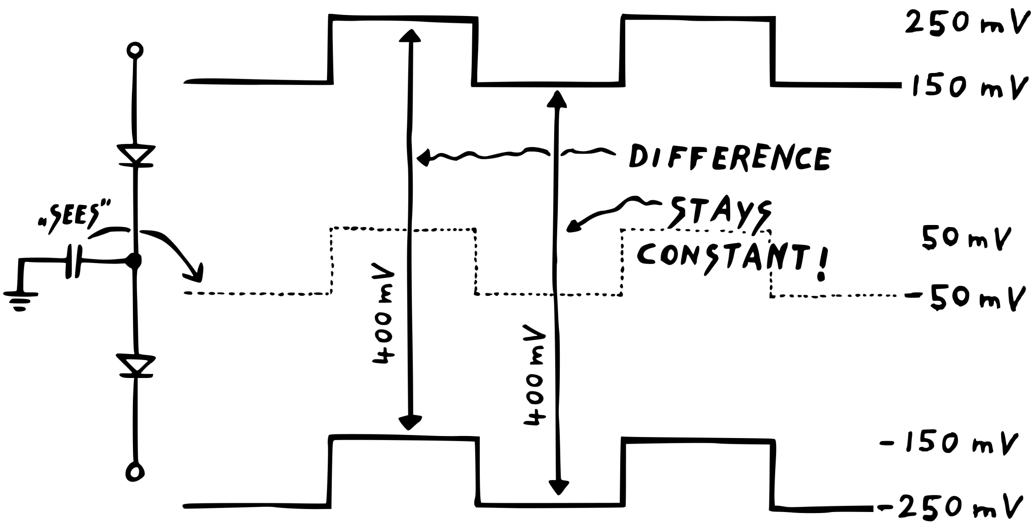


Because in order for this to work as a proper low-pass filter, these diodes' effective resistance values would need to be stable. But as we discovered earlier, those resistance values are directly tied to the diodes' input voltages. **So as our input signal oscillates, our filter's cutoff point would be moving around wildly – giving us a really mangled output waveform.**

Okay, but is there any way around this? It seems like the oscillation itself is the problem. Because in order to stabilize and control the diodes' effective resistances, we'd need to stabilize their in- and output voltages. So – game over? Not quite. There's a really nifty trick we can apply here.

THE SIMPLEST DIODE LADDER FILTER

So imagine we take our input oscillation, and we create two copies. We'll push one of them up using an offset voltage, and we'll drag the other one down by the same margin. Then, we'll modify our circuit, applying the raised oscillation to the top diode and the lowered oscillation to the bottom diode.[18] Okay, but how does this help?



Well, if you look at the voltage graphs, you'll notice something interesting. While both voltages are still oscillating, the voltage *difference* between them is staying constant at all times. And since voltages are always just relative, that voltage difference is all that matters here. Because in effect, our diodes will see a stable stream of current going through them. You can also call that our bias current. **And if this bias current stays constant, the diodes' resistance values will stay constant as well**.

Better yet: those resistance values now depend on the offset voltage we add to and subtract from our oscillation. Because if we reduce that offset, we pull the cloned oscillations closer together, decreasing the bias current. And this will reduce our diodes' effective resistances. Okay, but what does that mean for our filtering capacitor? If we cancel out the oscillation like that, stabilizing the current flowing through the diodes, won't that mean that the draining and filling of the cap will cease as well?

Luckily for us, it won't. This is because our capacitor is tied to ground on one side, making 0 V its reference point. So when both oscillations are in their high phase, the voltage at the top diode will be more positive (in relation to 0 V) than the voltage at the

[18] You can try this chapter's circuits in a circuit simulator. I've already set them up for you right here: https://tinyurl.com/y62brums – you can change all values by double clicking on components.

bottom diode is negative. And vice versa during the low phases. **Now, the trick here is that our two diodes practically form a perfect 50% voltage divider, since they'll have nearly identical effective resistances.** And as we know, such a 50% voltage divider will slash the voltage between its two reference points in half.

You can then imagine the dotted line in the center of the illustration as the voltage that the capacitor sees: it's basically the exact mid point between the top and bottom voltages. **And this mid point shifts upwards and downwards as the signal oscillates, effectively pushing and pulling current through our diodes into and out of the capacitor**. Sounds great – but there are two things to look out for here.

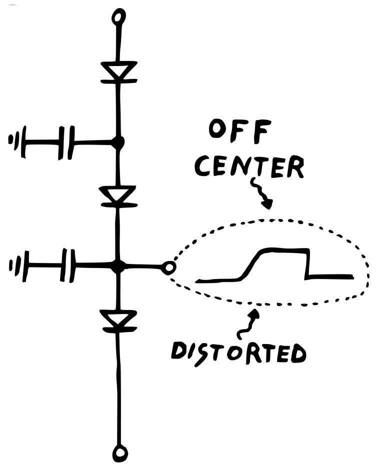
First of all, we should be wary of pushing the voltage difference between our two oscillations too much. This is because at some point, our diodes' effective resistances would get so low that we'd essentially create a short circuit.

And second, the oscillations themselves need to be kept very low in volume. The problem is that if the mid-point voltage swing gets too large, it will start messing with the bias current flowing through the diodes. This is because as it gets bigger, it pushes and pulls more and more current into and out of the capacitor. **A small mid-point swing will have a rather negligible effect on our bias current – but as you increase that swing, the effect will get pretty severe pretty quickly**. This, in turn, will alter our diode's effective resistances noticeably, resulting in an unstable cutoff point and a distorted output.

MULTI-STAGE DIODE LADDER

Okay, so with these things in mind, we would be able to build a basic single pole low pass filter. But what if we'd rather like to have a two- or a three pole version? Luckily for us, adding poles requires little to no extra effort. It's as simple as chaining multiple stages together.

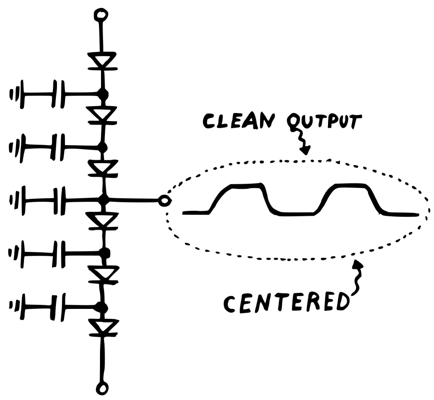
There is one small caveat, though: the total number of stages needs to be odd. This has two reasons which are closely interlinked. The first one is not crucial, but it's really a convenience-type thing. **If the total number of stages is odd, all the diodes taken together will form a big, evenly stepped voltage divider that has a single central node. And at that node, the voltage will always oscillate around 0 V**. Which will make it easy for us to grab the signal, because we won't have to remove an offset voltage.



The second reason is somewhat more critical, though. Imagine we design a filter with 2 stages. Then first, we'll have to decide from which node we'll pick up our output signal. Let's assume we choose the bottom one. Now we'll apply our raised square wave oscillation to the top, and our lowered counterpart to the bottom of the ladder.

During the high phases, the upper capacitor will be filled up first, simply because it's closer to the source. This means that until that cap is completely full, this second cap will only get the scraps that spill over. Effectively delaying the charging process and thereby the rise in voltage at the bottom node. So far, this is actually what we expect and want from a two-pole filter.

The actual problem arises during the oscillation's low phase. Because during that low phase, the bottom capacitor is closer to the drain. Which means that it now has „priority" and will get drained faster than the other cap. **This introduces a strong asymmetry into our output, because the falling edge will be steeper than the rising edge**.



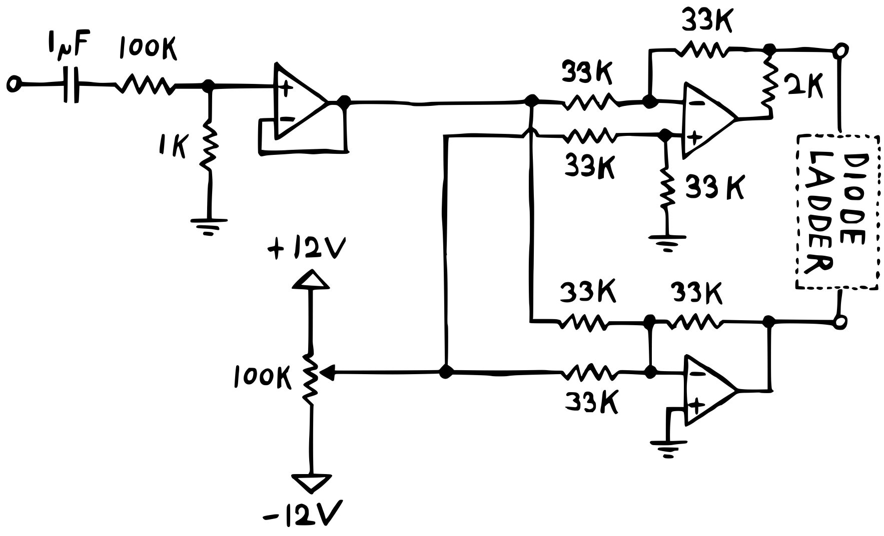
What can we do about it? Well, like I said before: we just need to make sure the total number of stages is odd. Because then, the capacitor at the middle node will always be as far from the source as it is from the drain, giving us a symmetrical, undistorted output wave. Which is also why a three-pole diode ladder filter consists of 5 individual stages. You can think of it like this (though it is an oversimplification): all the rising edges of our input wave will be filtered by the top three stages, while all the falling edges will be filtered by the bottom three.

Now, since there are no buffers between the stages, this is not entirely true, because all of them influence each other at any point in time. But as an approximation, I think we can go with this kind of model.

There is one final thing we need to talk about. You'll notice that I haven't drawn in any values for our capacitors here yet. So we'll have to decide on those before we can proceed. In my experiments, I've fared best with 1 nF caps, since those gave me a decent range – going from killing all frequencies to letting through up to 20 kHz.[19]

[19] You can try this chapter's circuits in a circuit simulator. I've already set them up for you right here: https://tinyurl.com/y5z9m64u – you can change all values by double clicking on components.

DRIVING THE LADDER

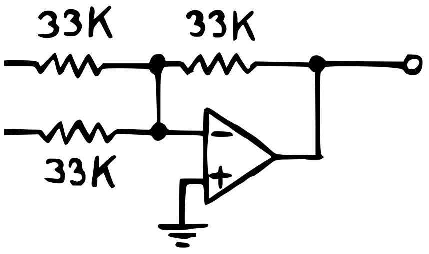


Okay, so now that we know how to properly set up our diode ladder, we'll need to think about how we'll feed it our input signal. Thankfully, doing that is pretty straightforward – even though this schematic might look a bit daunting at first. The basic idea is this. Using the top left input, we send in our oscillation. Below that, we have a potentiometer set up as a variable voltage divider. This will be our control voltage source for now – so that we can test every CV value between + and -12 V.

First of all, we'll send our input oscillation through AC coupling again to eliminate any offset voltage and center it around 0 V. Then, we'll shrink it down quite intensely using a 100k/1k voltage divider. I'm expecting the input signal to be 10 V peak-to-peak (which is the standard in eurorack systems), so this will give us a 100 mV peak-to-peak result. **Because remember, in order to not mess with the bias current too much later, our oscillation needs to be low in volume**. (You could get away with a bit more power, but I'm playing it fairly safe here. Feel free to experiment with louder input signals, though.) In order to make sure that the input signal is not loaded down by the rest of our circuit, we'll again buffer it with an op amp.

Next, we'll need to create two separate copies of our scaled-down oscillation and push one of them up – and the other one down using the control voltage created by the potentiometer. Now, there's a couple ways to do this – but I'm using op amps, since they're so easy to handle. Let's start with the lowered oscillation. While we could do this the „intuitive" way – which for me would mean first inverting the offset voltage, then mixing it into our oscillation and finally buffering the result – this would use up two op amps for something that you can essentially do with one. **The trick here is to use an inverting op amp buffer for both purposes at once.**[20] Here's how it works.

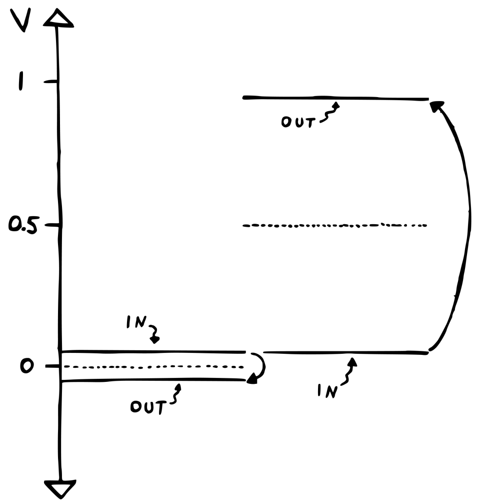
[20] Read more about the inverting op amp configuration in the components & concepts appendix (page 53).

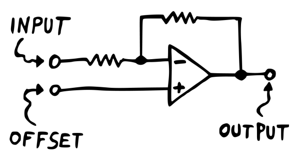


We connect our oscillation and the control voltage through resistors to an op amp's inverting input, while grounding the non-inverting input. Then, we add another resistor from inverting input to output. If the resistors are all of the same value – like they are here – you get the inverted sum of your inputs at the output.[21] So if we set our control voltage to, say, 1 V, the sum of that offset and our small oscillation would be a signal that's swinging from 1.05 V to 0.95 V. **Invert that, and we've successfully pushed down our oscillation by the given offset.**

Heads up, though: as a side-effect, we also inverted the oscillation itself. So we'll need to make sure that we invert the raised oscillation as well. Which is why we'll have to set up our second op amp in the inverting configuration also. But now the question is: how do we push our signal up instead of dragging it down?

The solution might confuse you at first. Because normally, when we set up an inverting amplifier, we connect the op amp's non-inverting input straight to ground. But if you check the complete circuit schematic again, you'll see that we feed the upper op amp's non-inverting input exactly half of our control voltage – hence the 50% voltage divider there. Why? **Well, the basic idea is that we want to push the op amp's point of reference upwards.**

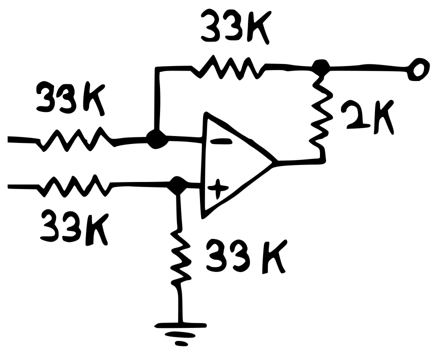




When the non-inverting input is connected to 0 V, the amplifier will invert the signal at the inverting input in relation to those 0 V. Which is why an input of 50 mV will see an output of -50 mV: 0 V is basically the mid axis across which we're jumping. By pushing that mid axis up, we pretty much recalibrate the output. So let's assume we apply 500 mV to the

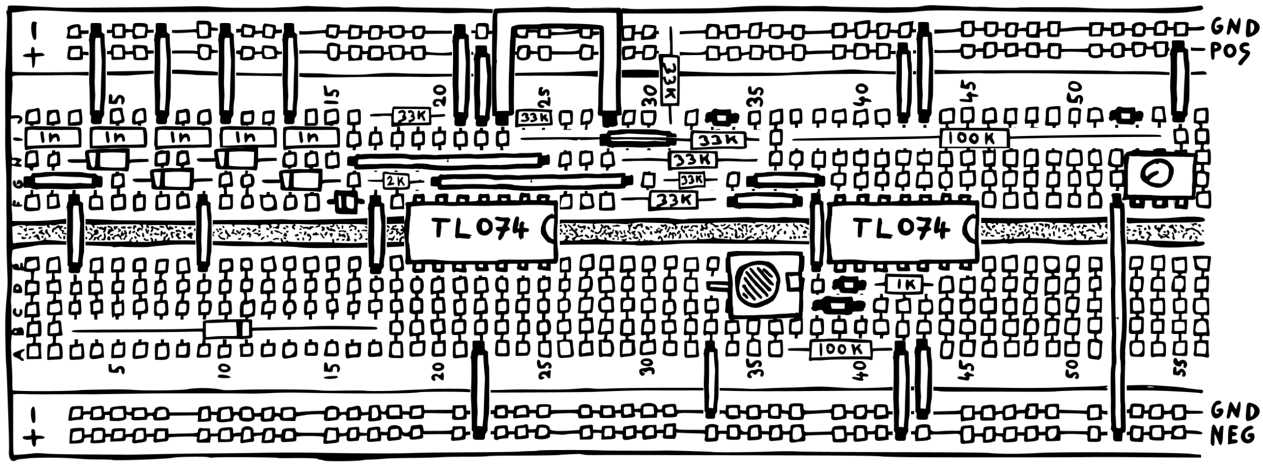
[21] A common question I get about this this setup is: why 33k resistors? Why not 100k resistors, which I normally use? Simple: to reduce the amount of noise added by our filter. Bigger resistors will naturally introduce more noise – which is especially problematic here because the signal going through our filter is so low in volume. The only downside is that we use a bit more current.

non-inverting input. Now, if we take our 50 mV and we jump that new mid axis, we get 950 mV as an output. And for -50 mV, we'd get 1.05 V.



So we've essentially inverted our oscillation while also pushing it up by the example control voltage – namely 1 V – through applying half that control voltage to the non-inverting input. Which is exactly what we're doing here with the 50% voltage divider at the non-inverting input.

Okay, so we've almost made our way through this maze of a circuit. There's just one small curiosity left: the 2k resistor after this op amp's output. Technically, it's not really necessary – it doesn't alter the voltage we feed into the ladder. But it is a protective measure.
Because as I explained before, if we push our diode ladder too far – meaning that we increase the voltage difference between the top and bottom op amps by too much –, the diodes' resistances will drop so low that we've essentially created a short circuit. **The 2k resistor will then ensure that we don't kill our op amps in the process, and it does that by limiting the maximum amount of current that flows through the ladder**. Cool! So let's finally start building this on the breadboard.[22]



Once you've set this up, you'll probably notice that there is no output to plug your headphones into yet. For now, you can only really check if the circuit is working as intended using an oscilloscope.[23] If you have one, attach its probe to the ladder's middle capacitor (which is technically the circuit's output) and play around with the control voltage setting.

---

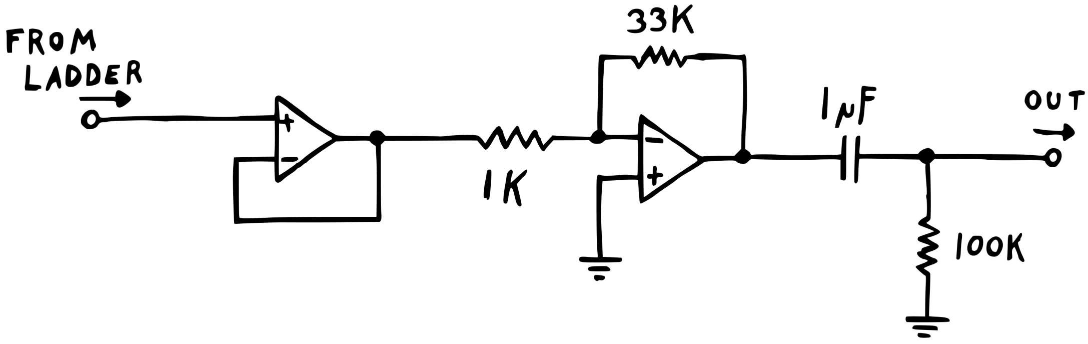
[22] You'll notice that I had to leave out the input AC coupling for space reasons. Also, the 2k resistor is at the bottom of the ladder, not at the top. It has the same effect as discussed above.

[23] Alternatively, you can try this chapter's circuits in a circuit simulator. I've already set them up for you right here: https://tinyurl.com/2alfoolb – you can change all values by double clicking on components.

THE OUTPUT STAGE

If you did try to look at the filtered signal, your oscilloscope probably had trouble tracing it through the noise. That's because it is so low in volume. After all, we shrunk the input oscillation down to just 100 mV peak-to-peak before sending it through our ladder. So if we really want to see what's going on (and/or listen to it), we need to amplify the signal by quite a lot.

Now, doing that is thankfully really straightforward. If you wanted to do it the super easy way, you could just set up a non-inverting amplifier with lots of gain and be done with it. But we should go the extra mile here and implement a slightly more complex solution.

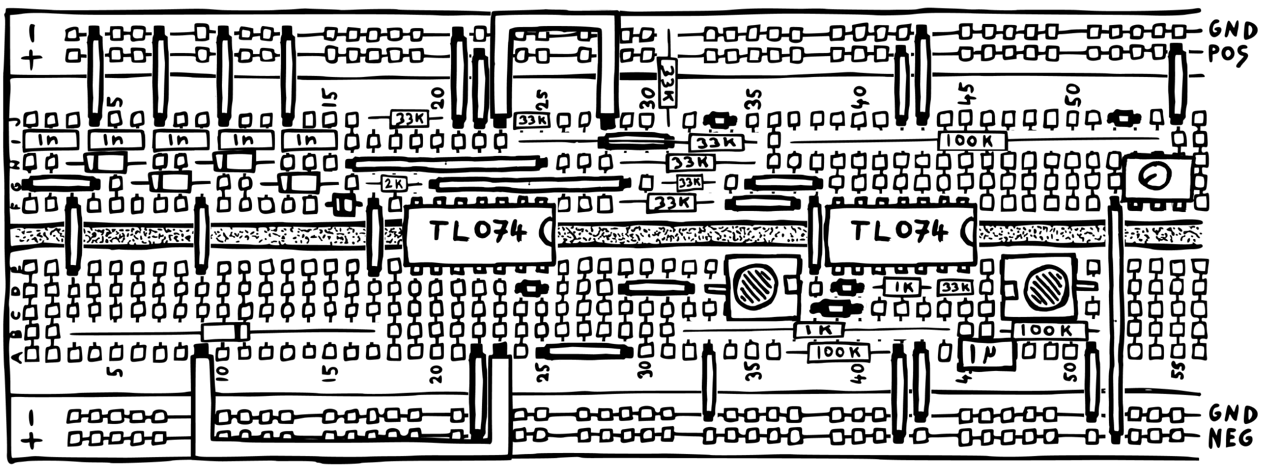


As you can see, I chose to first buffer the diode ladder's output before then blowing it up using an inverting amplifier. This has two reasons. **First of all, by inverting the output, we make sure that it's in phase with the input**. Because remember, while creating our two offset copies, we inverted both of them. So this pretty much reverses that. But in order to not load the ladder's output down with the inverting op amp, we need the buffer here. **And second, we will also need that buffer to implement the resonance path later**.

For setting the amplifier's gain, I'm picking rather conservative resistor values. These will get our signal up to around 3 V peak-to-peak and leave us with plenty of headroom for the aforementioned resonance. And as usual, I'll add AC coupling to the output in order to make sure that it's properly centered around 0 V.[24]

---

[24] You can try this chapter's circuit in a circuit simulator. I've already set it up for you right here: https://tinyurl.com/2yowgjq8 – you can change all values by double clicking on components.

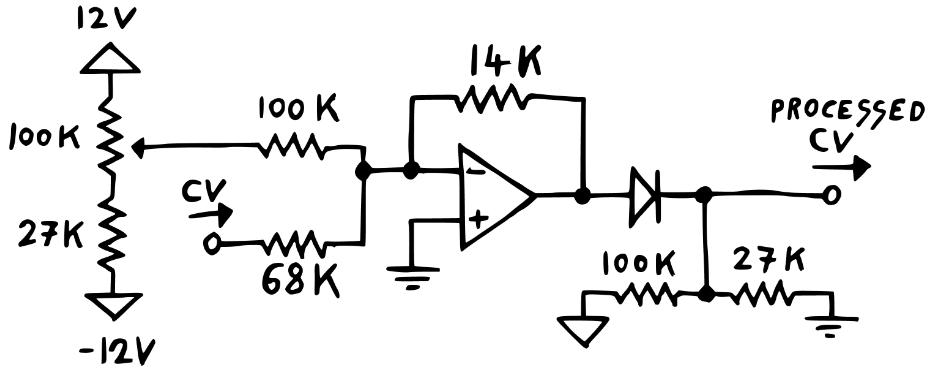


If you now check this on the oscilloscope, you should be able to make out the output oscillation much more clearly. Plug in your headphones and see if you get any sound out of it! If you don’t, try messing with the control voltage. **You’ll probably notice that the usable range is really small and therefore the knob is really finicky.** Also, if you turn it up too much, the signal will drop out completely.

This happens because we are pushing the bias current going through our diodes so much that their resistance values drop dramatically. Which causes all kinds of undesirable effects, essentially killing the oscillation at our filter’s output. What can we do about those two problems?

CV PROCESSING

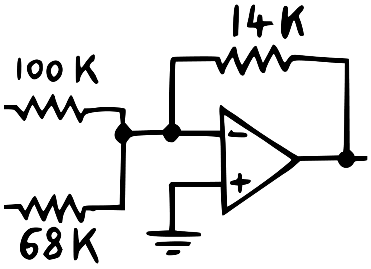
The answer is thankfully quite straightforward. We need to properly scale and also set an upper limit for our CV. While we're at it, we'll probably want to think about mixing multiple CV sources together, so that we can have a cutoff knob *as well as* an external CV input. Also, we should fine-tune that cutoff knob's range.



First, let's talk about the latter. As you can see, I simply added a 27k resistor between our potentiometer and the negative rail. This way, we raise its minimum output voltage from -12 V to just around -6 V. **The reasoning behind this is that at 0 V control voltage, there is no current going through the ladder (since the offset between our two oscillation copies is 0 V), and so the filter was completely closed**.

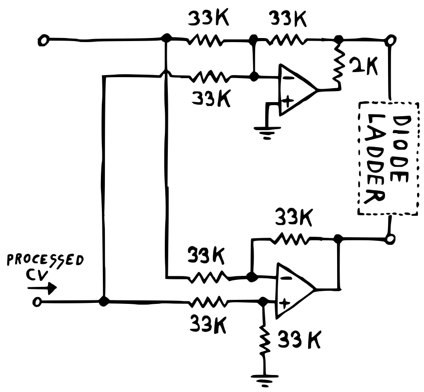
At control voltages below 0 V, the op amps were actually trying to push a current through the ladder in the opposite direction – which the diodes, being diodes, of course blocked. So the filter simply stayed closed. Now, you might ask why we would even have our cutoff knob send out voltages below 0 V, considering that this will result in a sizable dead zone in its range. But this is exactly the point.

Because the cutoff knob is no longer the only CV source. We've also got an external control voltage now that gets added to whatever the cutoff knob is currently sending out. **So the latter is basically setting a default cutoff frequency that the external CV is able to modulate**. And having a default cutoff frequency „below 0 Hz" is especially useful for short, plucky sounds. Of course we don't want the dead zone to occupy half of the cutoff knob's range – as it did before. Which is why I decided to shrink it down to just about one quarter using the 27k resistor.

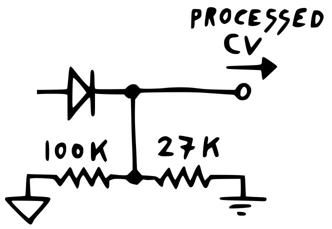


Okay, one down. Let's tackle the CV summing and scaling next. We'll do both with another op amp set up in the inverting amplifier configuration. Why inverting? **Because unlike the non-inverting configuration, this lets us add our input signals together without having them interact with each other in undesirable ways**. Also, scaling both the individual signals and the summed output down is much simpler this way.

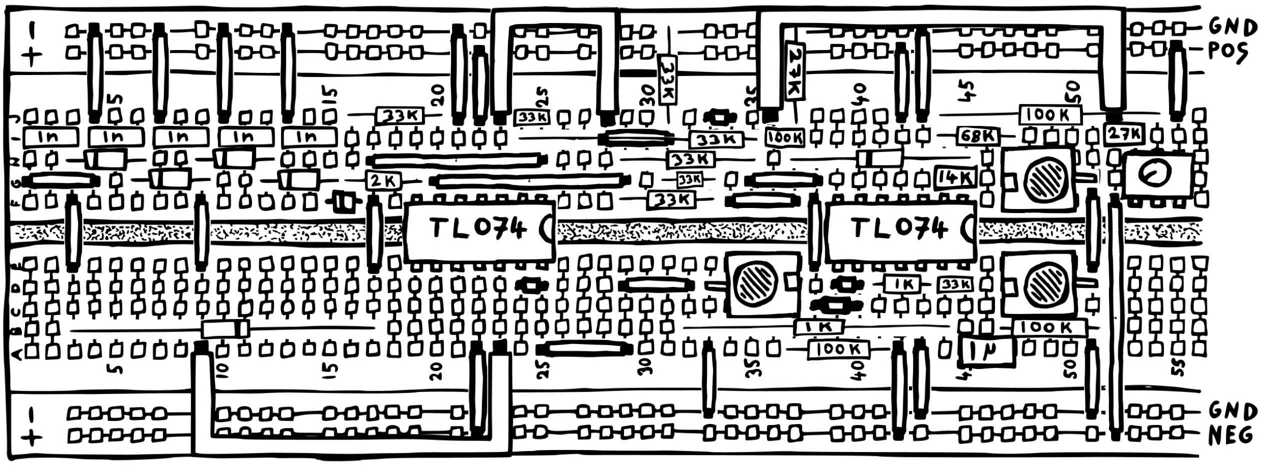
We just have to play around with the relation between the input- and feedback resistances. The bigger the input resistance compared to the feedback resistance, the more that input signal will be scaled down. As you can see, I've decided to shrink down our cutoff knob's voltage a lot more than the external CV. **This makes sense, because that cutoff knob maxes out at +12 V – while we can expect a typical external CV source (like an envelope generator) to peak at just around 8 V**. So the different resistor values should put them on equal footing.
Since both input resistors are still pretty huge compared to the feedback resistor, our summed output will be fairly small. This is important because as we saw before, the usable CV range for our diode ladder is fairly small as well. (Remember how finicky the cutoff knob was?) There is only one problem with this setup. Since we're using our op amp in the inverting configuration, it will of course invert our summed and scaled down CV.



This means that we can't simply send it to our ladder-driving op amps the same way we did before. Because remember – we set them up to expect a positive control voltage to cause a raise in cutoff frequency. Thankfully, we can reverse that pretty easily. **All we have to do is make the two amplifiers switch positions**. Then, the top op amp will push the oscillation up when given a negative control voltage – and the bottom one will pull it down. Problem solved! There is just one tiny thing left to talk about: setting an upper (or rather lower) limit for our processed CV.



Here's how we pull that off. We route our summing & scaling amplifier's output through a diode, which we then connect to a 27k/100k voltage divider between ground and the negative rail. This way, current can only flow through the diode if the voltage on the left is higher than the voltage on the right. Which, when connected to our ladder driving op amps, will be about -1.3 V. **Meaning that we can only raise the CV above that level – we can't lower it, because the diode is blocking that**. Why -1.3 V? Because this is right before our diode ladder starts glitching and killing the output signal.

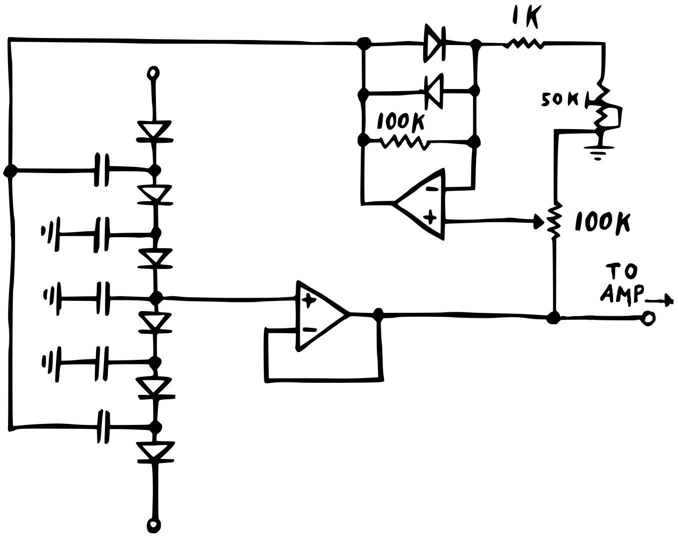


Once you plug your oscillator into the audio input, test the potentiometer's sweep without any external CV applied.[25] You should be getting a really decent, smooth response. **This is because our diode ladder is reacting in an exponential way to a linear increase in control voltage.** And since our ears basically work on a logarithmic scale, this sounds smooth and natural to us.

Next, plug in an LFO or envelope as an external CV source. The threshold-setting diode should hopefully be doing its job – so you shouldn't get any unexpected glitching behavior. If so: great! Next, we have one final, important thing to implement – resonance.

[25] You can also try this chapter's circuit in a circuit simulator. I've already set it up for you right here: https://tinyurl.com/2xp8ey77 – you can change all values by double clicking on components.

RESONANCE II

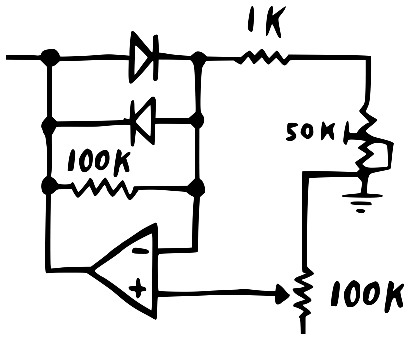


If you think back to our prior discussion about resonance, you might notice that the approach here is actually pretty similar. We're taking our filter's buffered output, routing it through a variable voltage divider and then amplifying it using yet another op amp. Finally, we're feeding the resulting signal – which can now be varied from silent to pretty intense – back into our filter through the top and bottom capacitor.

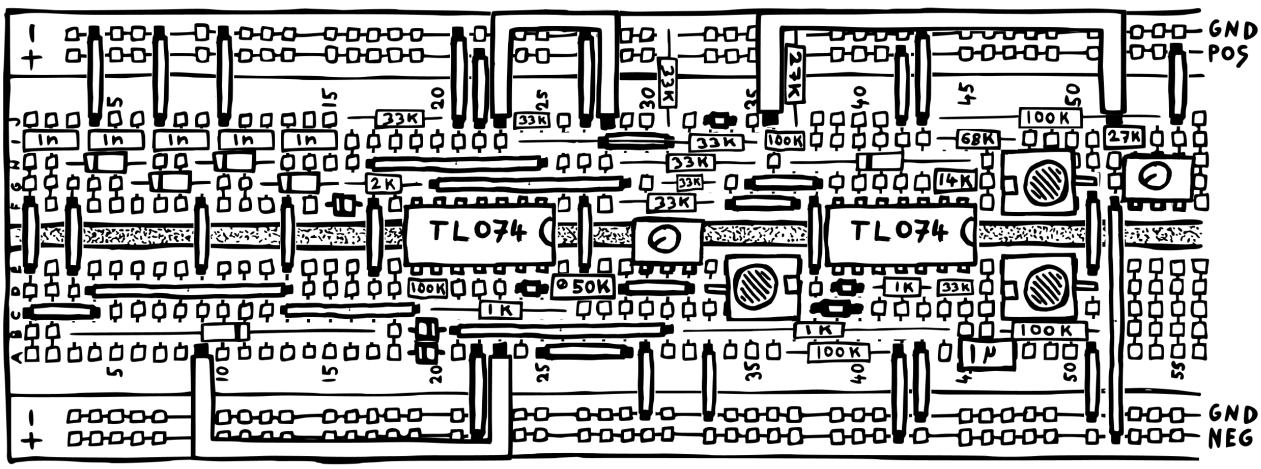
Now you might ask: why feed it into *both* of those? Well, this is simply due to the way we constructed our diode ladder. Remember that we are applying our input oscillation up top as well as down below in order to keep the bias current going through the diodes stable. **So if we only send our feedback into one of the capacitors, we would create an asymmetry that would mess with the filter's operation**. Sending it into both caps will then allow us to preserve that symmetry, because the top and bottom oscillations will be identical again – save for the offset.

Now, before we try this, let's talk about the op amp's configuration. Compared to the way we set it up previously, there's a bit more going on here. First, I added two diodes into the op amp's feedback path. These are here to clean up and control the resonance. How does this work? Simple. Remember how we said that our diode ladder will react exponentially to any linear increase in voltage across it? Well, now imagine that we remove these two feedback diodes. **This means that our feedback amplifier would operate in a strictly linear manner: so that any and all increases in volume at the filter's output would get amplified by a fixed factor and then fed back into the diode ladder**.

You can probably see how that would spiral out of control quite quickly: because there is nothing keeping the amount of feedback in check, it will begin to ramp up exponentially, until the top and bottom oscillations get so strong that they overpower the bias current and make our diode ladder shut down completely. This is where the two diodes come into play.



If we put them into the op amp's feedback path like shown here, they will essentially counteract the diode ladder's exponential behavior. Because as long as the filter's output is relatively quiet, our feedback diodes' effective resistances will be really high – which means that the op amp's gain is unaffected.
But as soon as our feedback loop begins its upward spiral, those diodes will respond by conducting more and more, lowering the amplifier's gain quite drastically. So in effect, we're basically straightening out the exponential curve here. Which will give us a silky smooth, clean resonance – if the feedback amplifier's base gain level is tuned correctly.
Because as a final touch, I've replaced the resistor between the op amp's inverting input and ground with a fixed 1k resistor and an additional 50k trimmer potentiometer. This allows us to experiment with different base gain levels. **The idea here is that the more base gain we dial in, the more the diodes in the feedback path are going to be activated.** When you push this past a certain threshold, those diodes are going to start warping the feedback in strange ways, resulting in a much more gritty-sounding resonance. Cool! So let's take to the breadboard one last time and try this out.[26]

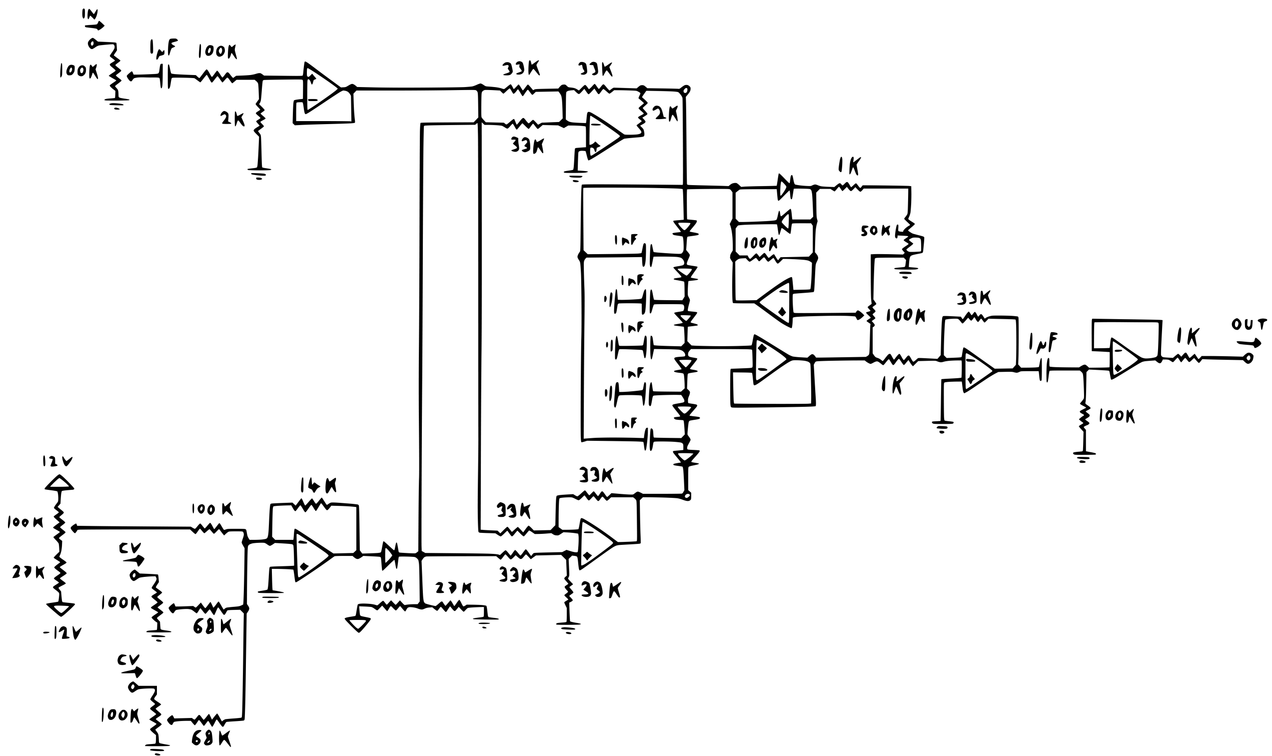


Send your oscillator into the audio input and play around with the resonance control knob. Depending on your trimmer's initial setting, you might not get any resonance at all. If so, you'll have to experiment by dialing in different values here. **Be aware that this is a precision trimmer – which means that you'll have to rotate the screw multiple times to change the feedback amplifier's base gain level noticeably**.

[26] You can also try this chapter's circuit in a circuit simulator. I've already set it up for you right here: https://tinyurl.com/23mp8quc – you can change all values by double clicking on components.

FINAL TOUCHES

Even though our filter is now fully functional, there are a couple extra things we've implemented for the production version.



First, we've added a second CV input. Thankfully, this is as simple as routing it into our CV summing amplifier through another 68k resistor. Then, we've added three 100k potentiometers set up as variable voltage dividers to both CV inputs and the audio signal input. **This allows us to adjust both the CV intensities and the signal input level on the fly**.

To make the latter more interesting, we've also replaced the 1k resistor going to ground before our signal input buffer with a 2k resistor. This raises the maximum signal input level by 50%, allowing us to dial in some filter drive. **Because remember: if the input signal gets louder, the diodes in our ladder will begin to distort the signal**.

Finally, we've added another op amp buffer to the output. This is not necessary and doesn't add anything to our circuit. But unfortunately, we're using exactly 7 op amps for our filter – which means that there is an unused one in our second TL074 chip. **Letting an op amp sit there idly without connecting it to anything is bad practice in a module build, because it will waste current, introduce crosstalk/noise and could potentially even damage the chip**. Setting it up as a buffer alleviates all of those problems – even if the buffer is unnecessary.

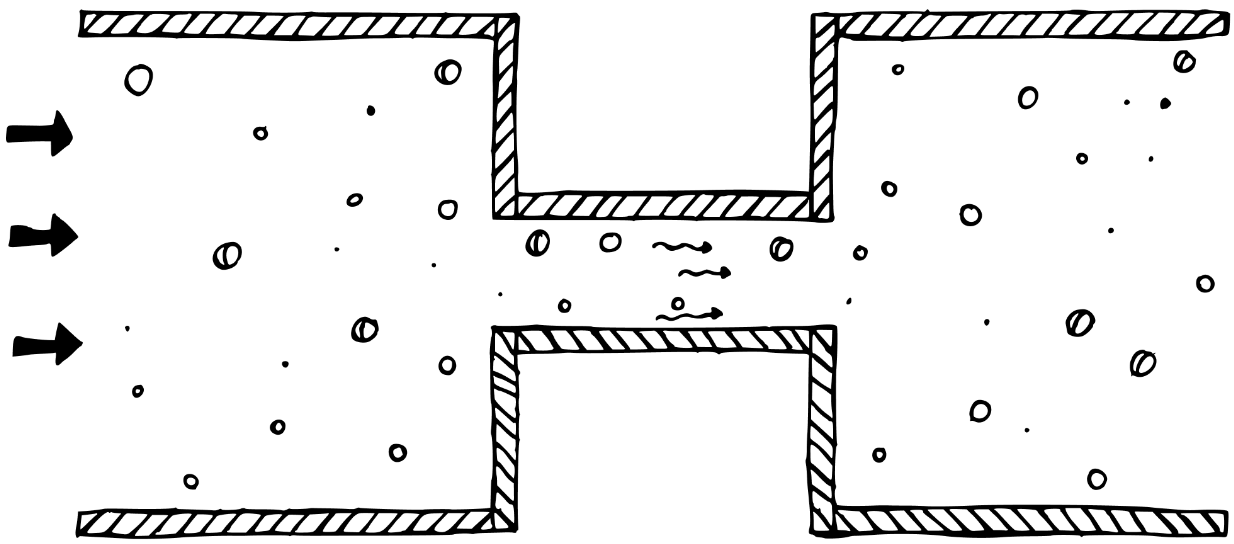
Now, because all of these additions are just refinements of features we already tried – and since our breadboard is already getting quite crowded – I'd say we can skip setting them up. Still, feel free to work them into your breadboard layout yourself!
And with that, our filter is done. If you now want to make your creation permanent, dig out the panel and PCB from the kit, heat up your soldering iron and get to building! You can find more information on how to populate the board & how to solder in the enclosed appendix.

COMPONENTS & CONCEPTS APPENDIX

In this section, we'll take a closer look at the components and elemental circuit design concepts we're using to build our module. Check these whenever the main manual moves a bit too fast for you!

THE BASICS: RESISTANCE, VOLTAGE, CURRENT

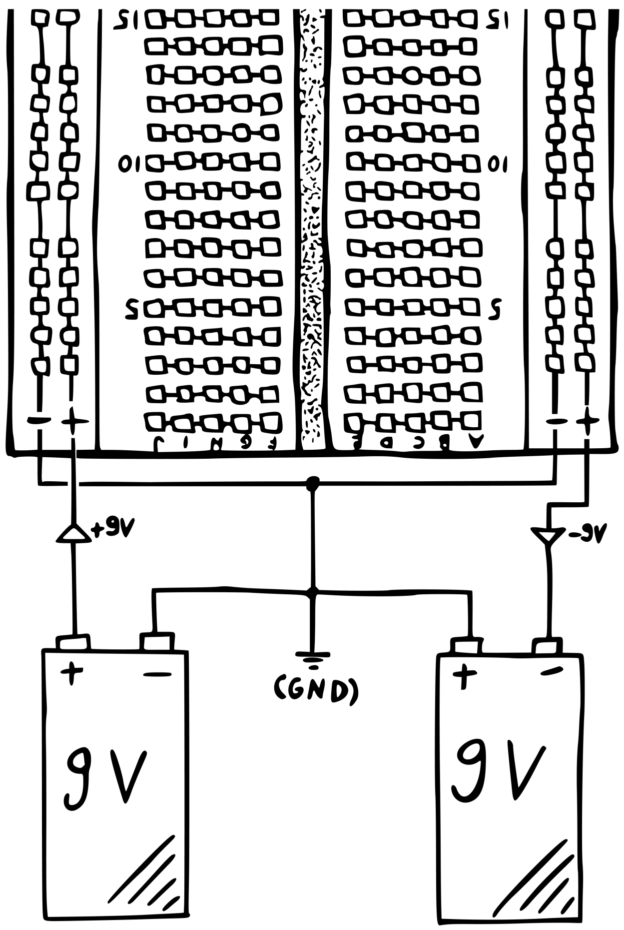
There are three main properties we're interested in when talking about electronic circuits: resistance, voltage and current. To make these less abstract, we can use a common beginner's metaphor and compare the flow of electrons to the flow of water through a pipe.



In that metaphor, resistance would be the width of a pipe. The wider it is, the more water can travel through it at once, and the easier it is to push a set amount from one end to the other. Current would then describe the flow, while voltage would describe the pressure pushing the water through the pipe. You can probably see how all three properties are interlinked: **more voltage increases the current, while more resistance to that voltage in turn decreases the current.**

USING TWO 9 V BATTERIES AS A DUAL POWER SUPPLY

Dual power supplies are great – and if you want to get serious about synth design, you should invest in one at some point. But what if you're just starting out, and you'd like to use batteries instead? Thankfully that's totally doable. **You just need to connect two 9 V batteries like shown here.** For this, you should use 9 V battery clips, which are cheap & widely available in every electronics shop.

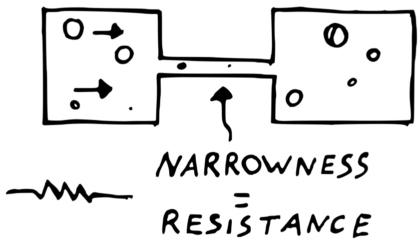


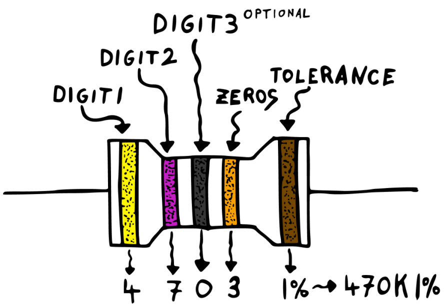
By connecting the batteries like this, the positive terminal of the left battery becomes your +9 V, while the negative terminal of the right is now your –9 V, and the other two combine to become your new ground.[27] **Please make sure you disconnect the batteries from your breadboard when you make changes to the circuit!** Otherwise you run the risk of damaging components.

[27] If you're struggling with setting this up, you can watch me do it here: https://youtu.be/XpMZoR3fgd0?t=742

RESISTORS

While a conductive wire is like a very big pipe where lots of water can pass through, **a resistor is like a narrow pipe that restricts the amount of water that can flow**. The narrowness of that pipe is equivalent to the resistance value, measured in ohms (Ω). The higher that value, the tighter the pipe.





Resistors have two distinctive properties: linearity and symmetry. Linearity, in this context, means that for a doubling in voltage, the current flowing will double as well. Symmetry means that the direction of flow doesn't matter – resistors work the same either way.

On a real-life resistor, you'll notice that its value is not printed on the outside – like it is with other components. Instead, it is indicated by colored stripes[28] – along with the resistor's tolerance rating. In addition to that, the resistor itself is also colored. Sometimes, depending on who made the resistor, this will be an additional tolerance indicator.

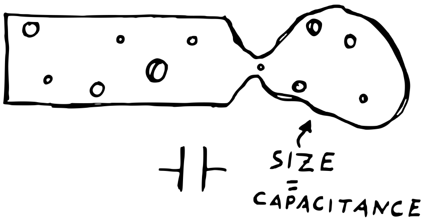
For the resistors in this kit, a yellow body tells you that the actual resistance value might be ±5 % off. A dark blue body indicates ±1 % tolerance. Some kits will also contain light blue ± 0.1% resistors to avoid the need for manual resistor matching.

While in the long run, learning all these color codes will be quite helpful, you can also simply use a multimeter to determine a resistor's value.

[28] For a detailed breakdown, look up resistor color coding. There are also calculation tools available.

CAPACITORS

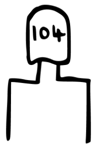
A capacitor is a bit like a balloon that you can attach to the open end of a pipe. If there's some pressure in the pipe, the balloon will fill up with water until the pressure equalizes. (Since the balloon needs some space to expand into, both of the capacitor's legs need to be connected to points in your circuit.)



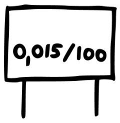
Then, should the pressure in the pipe drop, the balloon releases the water it stored into the pipe. The maximum size of the balloon is determined by the capacitor's capacitance, which we measure in farad (F). There are quite a few different types of capacitors: electrolytic, foil, ceramic, tantalum etc. They all have their unique properties and ideal usage scenarios – but the most important distinction is if they are polarized or not.

You shouldn't use polarized capacitors against their polarization (applying a negative voltage to their positive terminal and vice versa) – so they're out for most audio-related uses like AC coupling, high- & low-pass filters etc.

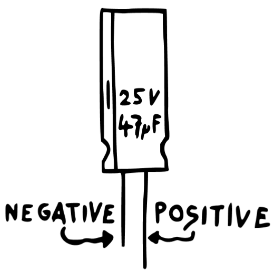
Unlike resistors, capacitors have their capacitance value printed onto their casing, sometimes together with a maximum operating voltage. **Be extra careful here!** That voltage rating is important. Your capacitors can actually explode if you exceed it! So they should be able to withstand the maximum voltage used in your circuit. If they're rated higher – even better, since it will increase their lifespan. No worries though: the capacitors in this kit are carefully chosen to work properly in this circuit.



Ceramic capacitors usually come in disk- or pillow-like cases, are non-polarized and typically encode their capacitance value.[29] Annoyingly, they rarely indicate their voltage rating – so you'll have to note it down when buying them.



Film capacitors come in rectangular, boxy cases, are non-polarized and sometimes, but not always, directly indicate their capacitance value and their voltage rating without any form of encoding.[30]



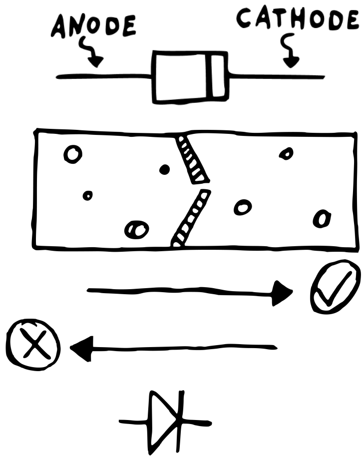
Electrolytic capacitors can be identified by their cylinder shape and silver top, and they usually directly indicate their capacitance value and their voltage rating. They are polarized – so make sure you put them into your circuit in the correct orientation.

---

[29] For a detailed breakdown, look up ceramic capacitor value code. There are also calculation tools available.

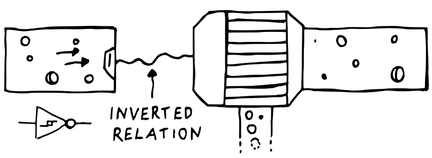
[30] If yours do encode their values, same idea applies here – look up film capacitor value code.

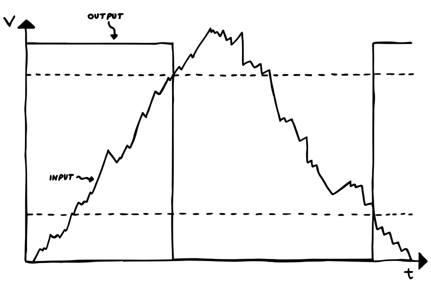
DIODES



Diodes are basically like one-way valves. Current can only pass through in one direction – from anode to cathode. That direction is indicated by the arrow in the diode symbol and by a black stripe on the diode's casing. So any current trying to move in the opposite direction is blocked from flowing.
There are a few quirks here, though. For one, the diode will only open up if the pushing force is strong enough. Generally, people say that's 0.7 V**,** but in reality, it's usually a bit lower. Also, diodes don't open up abruptly – they start conducting even at much lower voltages, although just slightly.
There are a lot of different diode types: Zener, Schottky, rectifier, small signal etc. They all have their unique properties and ideal usage scenarios – but usually, a generic 1N4148 small signal diode will get the job done.

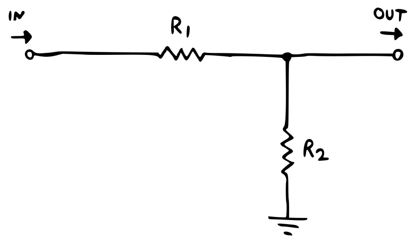
SCHMITT TRIGGER INVERTERS



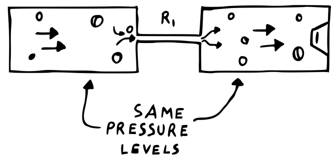


You can think of a Schmitt trigger inverter as two separate things. On the left, there's a sensor that measures the pressure inside an attached pipe. On the right, there is a water pump. This pump's operation is controlled by the sensor. Whenever the pressure probed by this sensor is below a certain threshold, the pump will be working. If the pressure is above a second threshold, the pump won't be working.
Here's a quick graph to visualize that. The squiggly line represents the voltage at the input, while the dotted line shows the voltage at the output. So every time we cross the upper threshold on our way up, and the lower one on our way down, the output changes its state. One thing that's very important to keep in mind: no current flows into the sensor! It's really just sensing the voltage without affecting it.

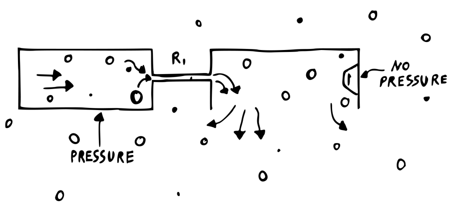
VOLTAGE DIVIDERS



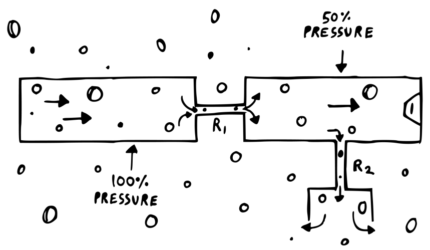
A voltage divider is really just two resistors set up like this: input on the left, output on the right. If R1 and R2 are of the same value, the output voltage will be half of what the input voltage is. How does it work?
Let's use our analogy again: so we have a pipe on the left, where water is being pushed to the right with a specific amount of force. Attached to it is a narrow pipe, representing R1, followed by another wide pipe. Then at the bottom, there's another narrow pipe, representing R2, where water can exit the pipe system. Finally, imagine we've set up a sensor measuring the voltage in the right hand pipe.



First, think about what would happen if R2 was completely sealed off. Our sensor would tell us that **the pressure on the right side is exactly the same as the pressure on the left**. Because the pushing force has nowhere else to go.
On the other hand, imagine R2 would just be a wide opening. Then **the pressure on the right would be 0**, because it'd all escape through that opening. But what happens if R2 is neither completely closed off nor wide open? Then the pressure would be retained to varying degrees, depending on the narrowness of the two resistor paths.

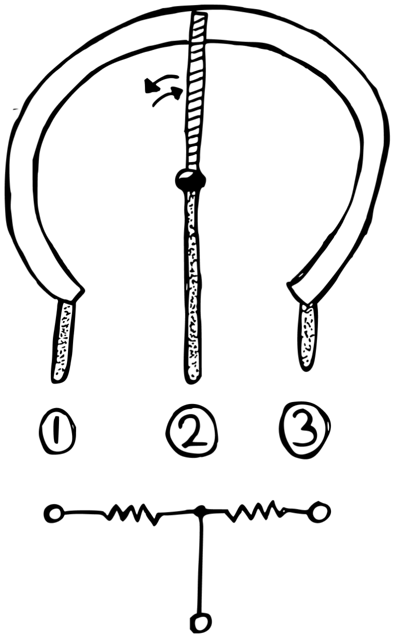


If pipe R1 is wide and pipe R2 is narrow, most of the pressure will be retained. But if it's the reverse, the pressure level will be only a tiny fraction. And if R1 and R2 are identical, **the pressure will be exactly half of what we send in**.



POTENTIOMETERS

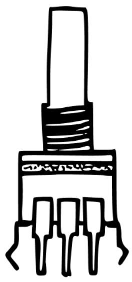
Potentiometers can be used as variable resistors that you control by turning a knob. But, and that's the handy part, they can also be set up as variable voltage dividers. To see how that works, let's imagine we open one up.

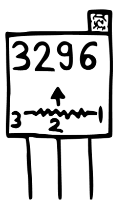


Inside, we would find two things: a round track of resistive material with connectors on both ends plus what's called a wiper. This wiper makes contact with the track and also has a connector. It can be moved to any position on the track. Now, the resistance value between the two track connectors is always going to stay exactly the same. That's why it's used to identify a potentiometer: as a 10k, 20k, 100k etc. But if you look at the resistance between either of those connectors and the wiper connector, you'll find that this is completely dependent on the wiper's position.

The logic here is really simple: **the closer the wiper is to a track connector, the lower the resistance is going to be between the two**. So if the wiper is dead in the middle, you'll have 50 % of the total resistance between each track connector and the wiper.

From here, you can move it in either direction and thereby shift the ratio between the two resistances to be whatever you want it to be. By now, you might be able to see how that relates to our voltage divider. If we send our input signal to connector 1 while grounding connector 3, we can pick up our output signal from the wiper. Then by turning the potentiometer's knob, we can adjust the voltage level from 0 to the input voltage – and anything in between.





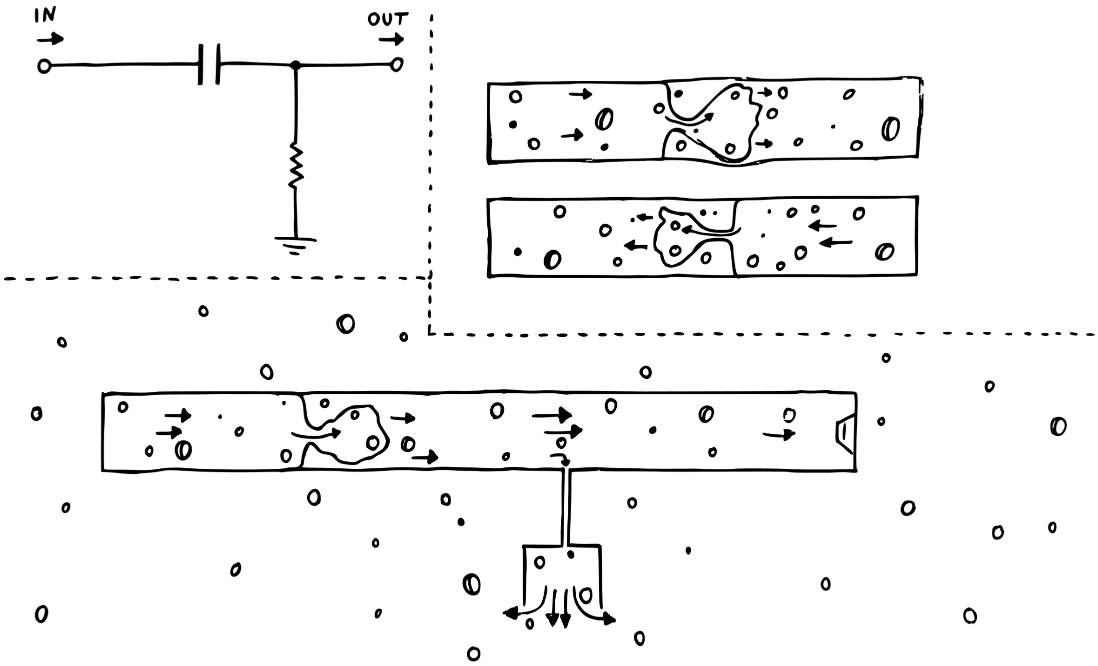
In these kits, you will encounter different types of potentiometers. First, there's the regular, full-size variant with a long shaft on top. These are used to implement user-facing controls on the module's panel and they usually – but not always – indicate their value directly on their casing. Sometimes, they'll use a similar encoding strategy as capacitors, though.[31]

Second, we've got the trimmer potentiometer, which is usually much smaller and doesn't sport a shaft on top. Instead, these have a small screw head which is supposed to be used for one-time set-and-forget calibrations. Trimmers usually encode their value.

[31] Look up potentiometer value code for a detailed breakdown.

AC COUPLING

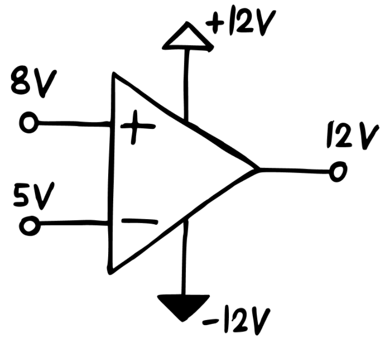
What is AC coupling – and how does it work? Imagine two adjacent pipes with a balloon between them. Now, no water can get from one pipe into the other, since it's blocked by the balloon. But, and that's the kicker, **water from one side can still push into the other by bending and stretching the balloon, causing a flow by displacement**.



Next, we'll bring in a resistor after the coupling point, going straight to ground. **This acts like a kind of equalizing valve**. Now imagine we apply a steady 5 V from one side. Then on the other side, we'll read 0 V after a short amount of time. Why? Because we're pushing water into the balloon with a constant force, causing it to stretch into the other side, displacing some water. If we didn't have the equalizing valve there, we'd simply raise the pressure. But since we do have it, the excess water can drain out of the system. Until the pressure is neutralized, and no water is actively flowing anymore.
Okay, so now imagine that the voltage on the left hand side starts oscillating, let's say between 4 V and 6 V. When we start to go below 5 V, the balloon will begin contracting, basically pulling the water to the left. This will create a negative voltage level in the right hand pipe – like as if you're sucking on a straw, making the voltage there drop below 0 V. Then, once the pressure on the other side rises above 5 V, the balloon will inflate and stretch out again, pushing water to the right. And the pressure in the right hand pipe will go positive, making the voltage rise above 0 V. **We've re-centered our oscillation around the 0 V line.** Okay, but what about the resistor? If current can escape through it, doesn't that mess with our oscillation? Well, technically yes, but practically, we're choosing a narrow enough pipe to make the effect on quick pressure changes negligible!

OP AMPS

Op amps might seem intimidating at first, but they're actually quite easy to understand and use. The basic concept is this: every op amp has two inputs and one output. Think of those inputs like voltage sensors. You can attach them to any point in your circuit and they will detect the voltage there without interfering. **No current flows into the op amps inputs – that's why we say their input impedance is very high**. Near infinite, actually. Okay, but why are there two of them?



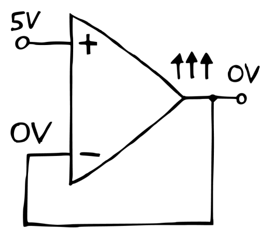
The key here is that op amps are essentially differential amplifiers. This means that they only amplify the difference between their two inputs – not each of them individually. If that sounds confusing, let's check out a quick example. So we'll imagine that one sensor – called the non-inverting input – is reading 8 V from somewhere. The other sensor – called the inverting input – reads 5 V. Then, as a first step, the op amp will subtract the inverting input's value from the non-inverting input's value. Leaving us with a result of 3. (Because 8 minus 5 is 3.) **This result then gets multiplied by a very large number – called the op amp's gain**. Finally, the op amp will try to push out a voltage that corresponds to that multiplication's result.

But of course, the op amp is limited here by the voltages that we supply it with. If we give it –12 V as a minimum and +12 V as a maximum, the highest it can go will be +12 V. So in our example, even though the result of that multiplication would be huge, the op amp will simply push out 12 V here and call it a day.

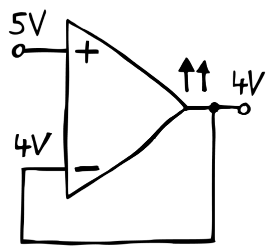
The handy thing though about op amp outputs is that they draw their power directly from the power source. This means that they can supply lots of current while keeping the voltage stable. **That's why we say an op amp has a very low output impedance**.

OP AMP BUFFERS/AMPLIFIERS

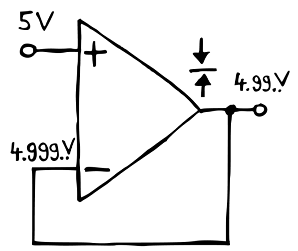
Buffering, in the world of electronics, means that we provide a perfect copy of a voltage without interfering with that voltage in the process. With an op amp-based buffer, the buffering process itself works like this. We use the non-inverting input to probe a voltage, while the inverting input connects straight to the op amp's output. **This creates what we call a negative feedback loop**. Think of it this way. We apply a specific voltage level to the non-inverting input – let's say 5 V.



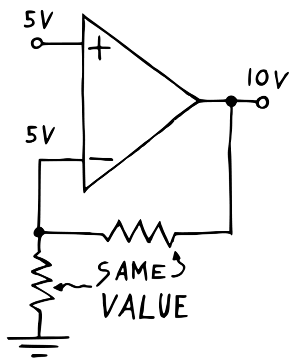
Before the op amp starts processing the voltages at its inputs, the output will be switched off. This means that **output and inverting input sit at 0 V at first**. So then, the op amp will subtract 0 from 5 and multiply the result by its gain. Finally, it will try and increase its output voltage to match the calculation's outcome.



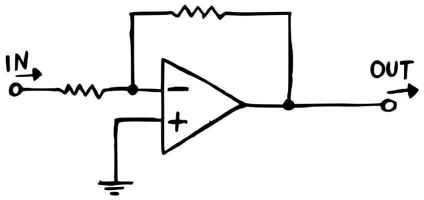
But as it's pushing up that output voltage, the **voltage at the inverting input will be raised simultaneously**. So the difference between the two inputs is shrinking down. Initially, this doesn't matter much because the gain is so large. As the voltage at the inverting input gets closer to 5 V though, the difference will shrink so much that in relation, the gain suddenly isn't so large anymore.



Then, the output will **stabilize at a voltage level that is a tiny bit below 5 V**, so that the difference between the two inputs multiplied by the huge gain gives us exactly that voltage slightly below 5 V. And this process simply loops forever, keeping everything stable through negative feedback. Now if the voltage at the non-inverting input changes, that feedback loop would ensure that the output voltage is always following. So that's why this configuration works as a buffer: the **output is simply following the input**.



How about amplifying a signal though? To do that, we'll have to turn our buffer into a proper non-inverting amplifier. We can do that by replacing the straight connection between inverting input and output with a voltage divider, forcing the op amp to work harder. Here's how that works. Say we feed our non-inverting input a voltage of 5 V. Now, **the output needs to push out 10 V in order to get the voltage at the inverting input up to 5 V**. We call this setup a non-inverting



amplifier because the output signal is in phase with the input.

For an inverting buffer/amplifier, the input signal is no longer applied to the non-inverting input. Instead, that input is tied directly to ground. So it'll just sit at 0 V the entire time. The real action, then, is happening at the inverting input. Here, we first send in our waveform through a resistor. Then, the inverting input is connected to the op amp's output through another resistor of the same value.

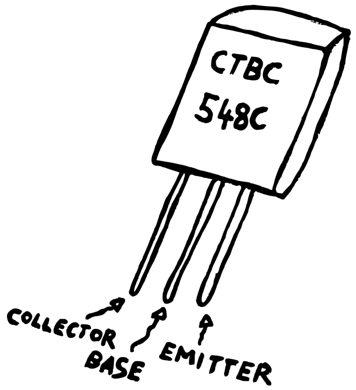
How does this work? Well, let's assume that we're applying a steady voltage of 5 V on the left. Then, as we already know, the op amp will subtract the inverting input's voltage from the non-inverting input's voltage, leaving us with a result of –5 V. Multiply that by the huge internal gain, and the op amp will try to massively decrease the voltage at its output.

But as it's doing that, an increasingly larger current will flow through both resistors and into the output. Now, as long as the pushing voltage on the left is stronger than the pulling voltage on the right, some potential (e.g. a non-zero voltage) will remain at the inverting input. Once the output reaches about –5 V though, we'll enter a state of balance. Since both resistors are of the same value, the pushing force on the left is fighting the exact same resistance as the pulling force on the right. **So all of the current being pushed through one resistor is instantly being pulled through the other**.

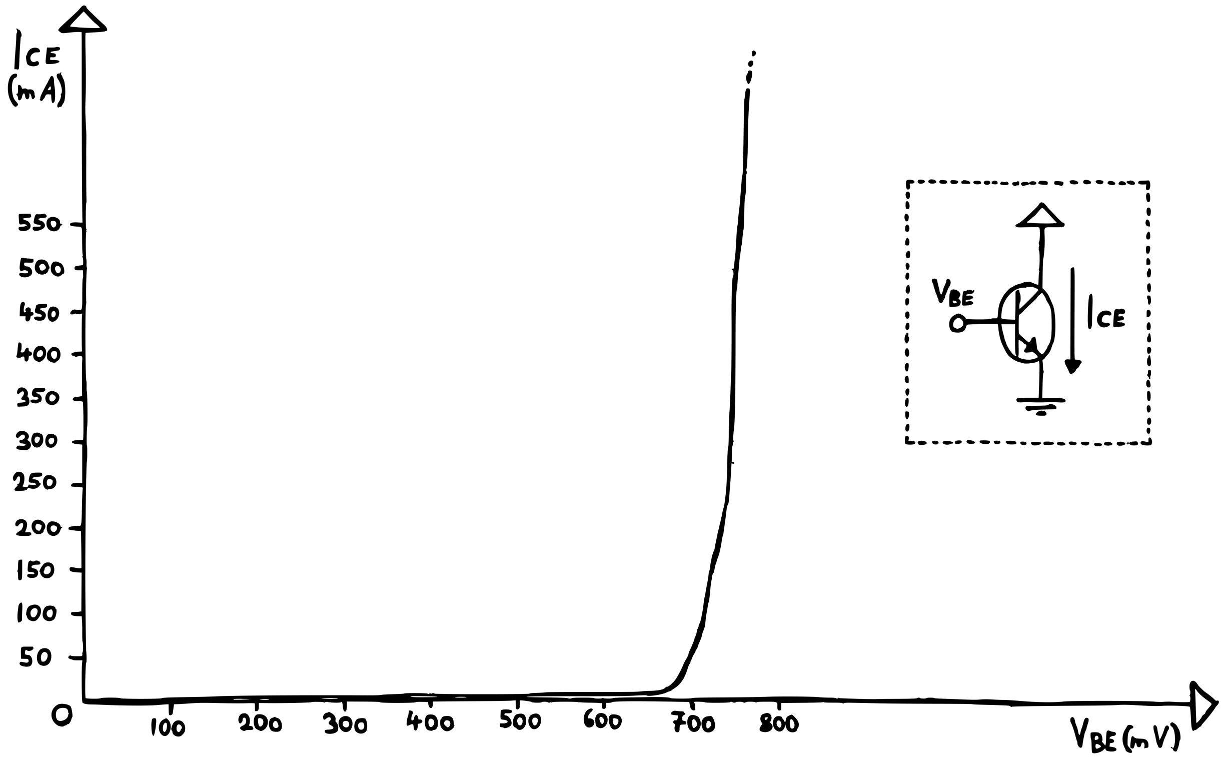
And that means that the voltage at the inverting input will be lowered to about 0 V, allowing our op-amp to settle on the current output voltage level. So while we read 5 V on the left, we'll now read a stable –5 V at the op amp's output. Congrats – we've built an inverting buffer! **If we want to turn it into a proper amplifier, we'll simply have to change the relation between the two resistances**. By doing this, we can either increase (if you increase the right-hand resistor's value) or reduce (if you increase the left-hand resistor's value) the gain to our heart's content.

BIPOLAR JUNCTION TRANSISTORS

Bipolar junction transistors (or BJTs for short) come in two flavors: NPN and PNP. This refers to how the device is built internally and how it'll behave in a circuit. Apart from that, they look pretty much identical: a small black half-cylinder with three legs.



Let's take a look at the more commonly used NPN variant first. Here's how we distinguish between its three legs. **There's a collector, a base and an emitter.**[32] All three serve a specific purpose, and the basic idea is that you control the current flow between collector and emitter by applying a small voltage[33] to the base. The relation is simple: **more base voltage equals more collector current**. Drop it down to 0 V and the transistor will be completely closed off. Sounds simple – but there are four important quirks to this.



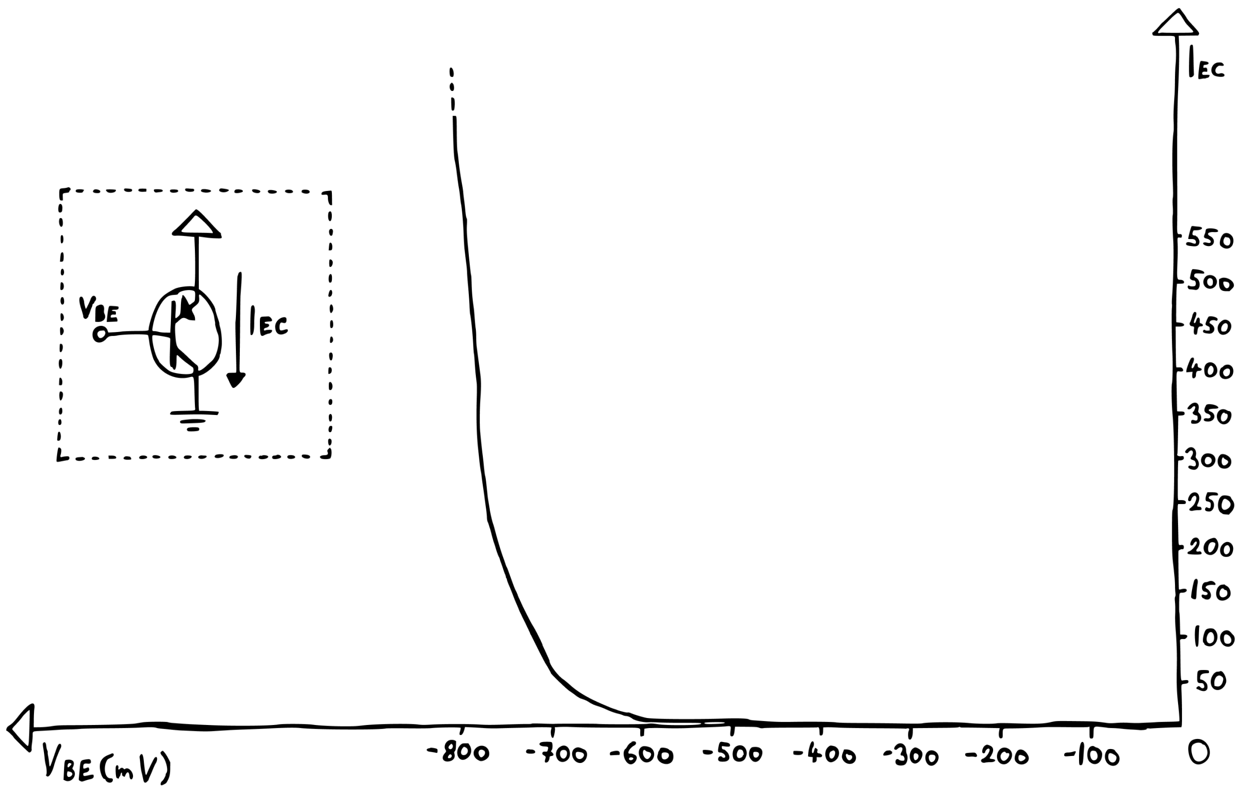
First, the relation between base voltage and collector current is exponential. Second, unlike a resistor, a BJT is not symmetrical – so we can't really reverse the direction of the

[32] Please note that the pinout shown here only applies for the BC series of transistors. Others, like the 2N series, allocate their pins differently.

[33] The voltage is measured between base and emitter. So „a small voltage“ effectively means a small voltage **difference** between base and emitter!

collector current. (At least not without some unwanted side effects.) Third, also unlike a resistor, a BJT is not a linear device. Meaning that a change in collector voltage will not affect the collector current. And fourth, the collector current is affected by the transistor's temperature! The more it heats up, the more current will flow.

Now, for the PNP transistor, all of the above applies, too – except for two little details. Unlike with the NPN, **the PNP transistor decreases its collector current when the voltage at its base increases**[34]. So you have to bring the base voltage below the emitter to open the transistor up. Also, that collector current flows out of, not into the collector!

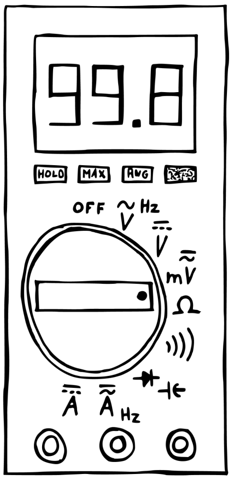


[34] Again, the voltage is measured between base and emitter.

TOOLS APPENDIX

There are two types of tools that will help you tremendously while designing a circuit: multimeters and oscilloscopes. In this appendix, we'll take a quick look at each of these and explore how to use them.

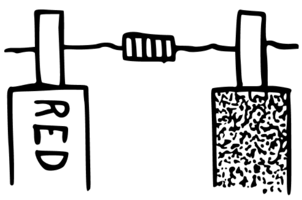
MULTIMETERS

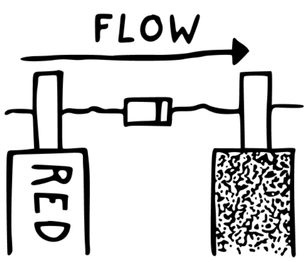


Multimeters come in different shapes and sizes, but the most common type is probably the hand-held, battery powered variant. It can measure a bunch of different things: voltage, current, resistance, continuity. Some have additional capabilities, allowing you to check capacitance, oscillation frequency or the forward voltage drop of a diode.

When shopping for one, you'll probably notice that there are really expensive models boasting about being TRUE RMS multimeters. For our purposes, this is really kind of irrelevant, so don't feel bad about going for a cheap model!

Using a multimeter is actually really straightforward. Simply attach two probes to your device – the one with a black cable traditionally plugs into the middle, while the red one goes into the right connector. Next, find whatever you want to measure and select the corresponding mode setting.





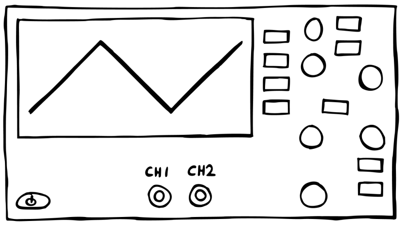
In some cases, it doesn't matter which probe you connect to which component leg or point in your circuit. This is true for testing resistors, non-polarized capacitors (foil/film, ceramic, teflon, glass etc.), continuity[35] or AC voltage.

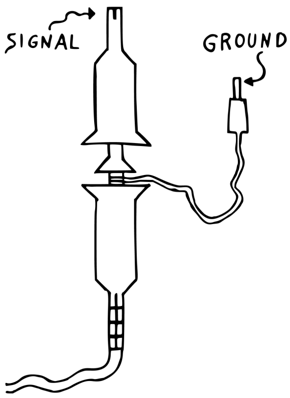
In others, you'll have to be careful about which probe you connect where. For testing the forward voltage drop of a diode, for example, **the multimeter tries to push a current from the red to the black probe**. Here, you'll have to make sure the diode is oriented correctly, so that it doesn't block that current from flowing. For testing a DC voltage, you want to make sure the black probe is connected to ground, while you use the red one to actually take your measurement.

---

[35] Just a fancy word for saying that two points are electrically connected.

OSCILLOSCOPES



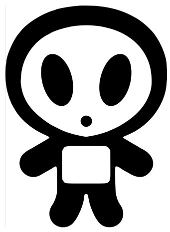


While multimeters are fairly cheap and compact, oscilloscopes are usually somewhat pricey and bulky. **If you're willing to make the investment, they are a huge help with the troubleshooting process, though**. Using one is, again, surprisingly straightforward – if you manage to work your way through the sometimes quite convoluted UI, especially on digital models.

To start using your scope, simply attach a probe to one of the channel inputs. These probes usually have two connectors on the other end: a big one that you operate by pulling the top part back – and a smaller one, which is usually a standard alligator clip. The latter needs to be connected to your circuit's ground rail, while you probe your oscillation with the former. Now what the oscilloscope will do is **monitor the voltage between the two connectors over time and draw it onto the screen as a graph**. Here, the x-axis is showing time, while the y-axis is showing voltage. You can use the device's scaling controls to zoom in on a specific part of your waveform.

Usually, digital oscilloscopes will also tell you a couple useful things about the signal you're currently viewing: minimum/maximum voltage level, oscillation frequency, signal offset. Some even offer a spectrum analyzer, which can be useful to check the frequencies contained in your signal.

BUILD GUIDE



MODULE ASSEMBLY APPENDIX

Before we start building, let's take a look at the complete **mki x es.edu Voltage Controlled Filter (VCF)** schematics (see next page) that were used for the final module's design and PCB fabrication. Most components on the production schematics have denominations (a name – like R1, C1, VT1, VD1, etc.) and values next to them. Denominations help identify each component on the PCB, which is particularly useful during **calibration, modification** or **troubleshooting.**

XS1 is the **Audio input** jack socket, **XS2 and XS3** are two identical **CV Input** jack sockets and **XS4** is the **Audio output** jack socket – these are the very same we've already been using on the breadboard for interfacing with other devices. In our designs, we use eurorack standard 3,5mm jack sockets (part number WQP-PJ301M-12).

XP1 is a standard eurorack **power connector.** It's a 2x5 male pin header with a key (the black plastic shroud around the pins) to prevent accidental reverse polarity power supply connection. This is necessary because connecting the power incorrectly will permanently damage the module.

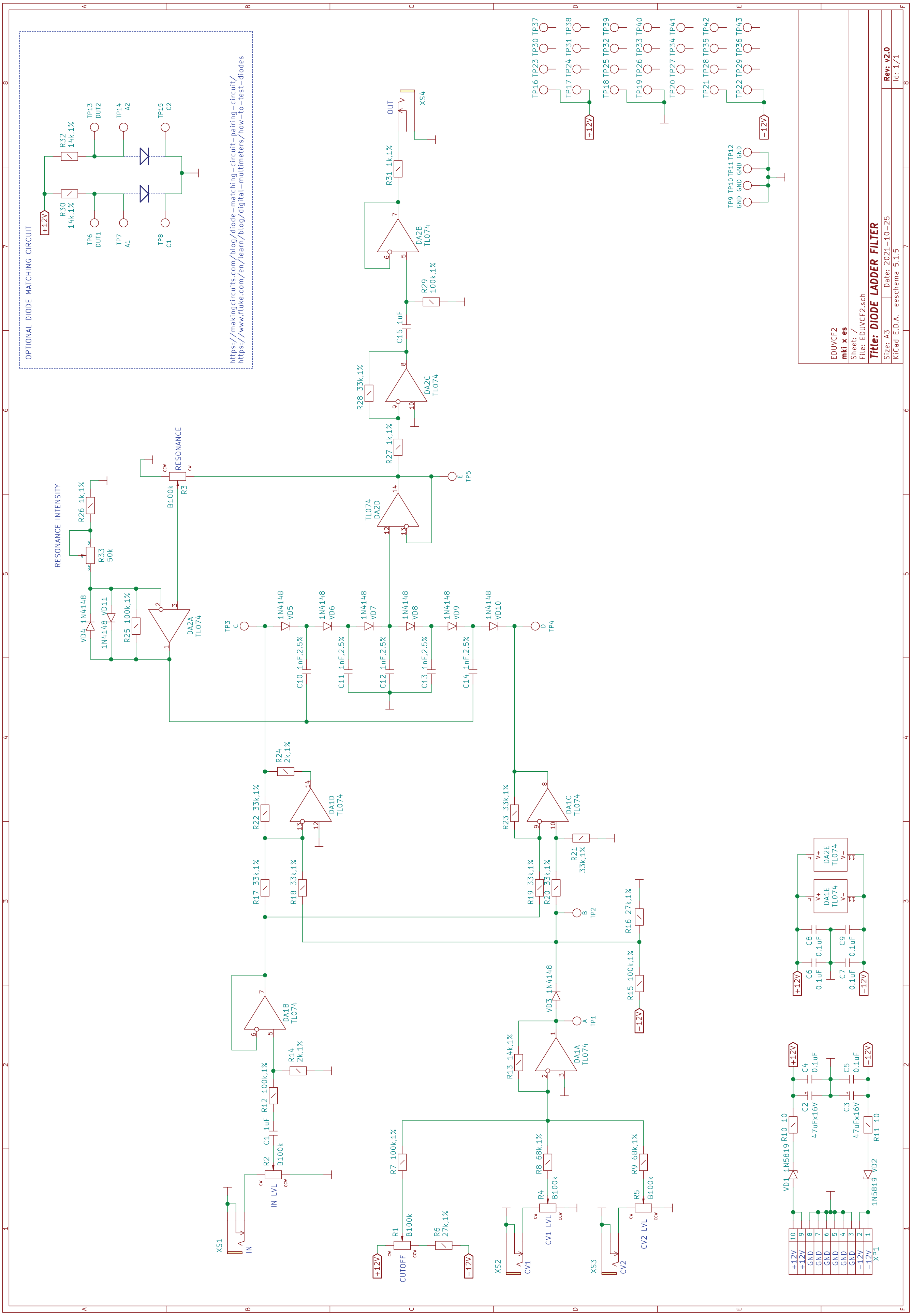
VD1 and **VD2** are **schottky diodes** that double-secure the reverse polarity power supply protection. Diodes pass current only in one direction. Because the anode of VD1 is connected to +12 V on our power header, it'll only conduct if the connector is plugged in correctly. If a negative voltage is accidentally applied to the anode of VD2, it closes, and no current passes through. The same goes for VD2, which is connected to -12 V. Because schottky diodes have a low forward voltage drop, they are the most efficient choice for applications like this.

Next, we have two **10 Ohm resistors (R10** and **R11)** on the + and – 12 V rails, with **decoupling** (or **bypass-**) capacitors C2 – C5. These capacitors serve as energy reservoirs that keep the module's internal supply voltages stable in case there are any fluctuations in the power supply of the entire modular system. In combination with R10 and R11, the large 47 microfarad pair (C2 and C3) compensates for low frequency fluctuations, while C4 and C5 filter out radio frequencies, high frequency spikes from switching power supplies and quick spikes created by other modules. Often another component – a **ferrite bead** – is used instead of a 10 Ohm resistor and there's no clear consensus among electronic designers which works best, but generally for analogue modules that work mostly in the audio frequency range (as opposed to digital ones that use microcontrollers running at 8 MHz frequencies and above), resistors are considered to be superior.

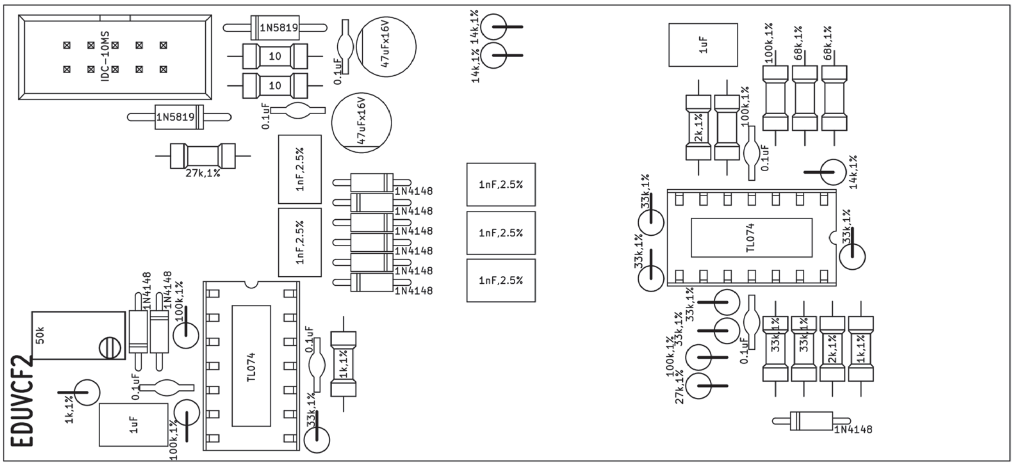
Another advantage of 10 Ohm resistors is that they will act like **slow "fuses"** in case there's an accidental short circuit somewhere on the PCB, or an integrated circuit (IC) is inserted backwards into a DIP socket. The resistor will get hot, begin smoking and finally break the connection. Even though they aren't really fuses, just having them there as fuse substitutes is pretty useful - **you'd rather lose a cent on a destroyed resistor than a few euros on destroyed ICs.**

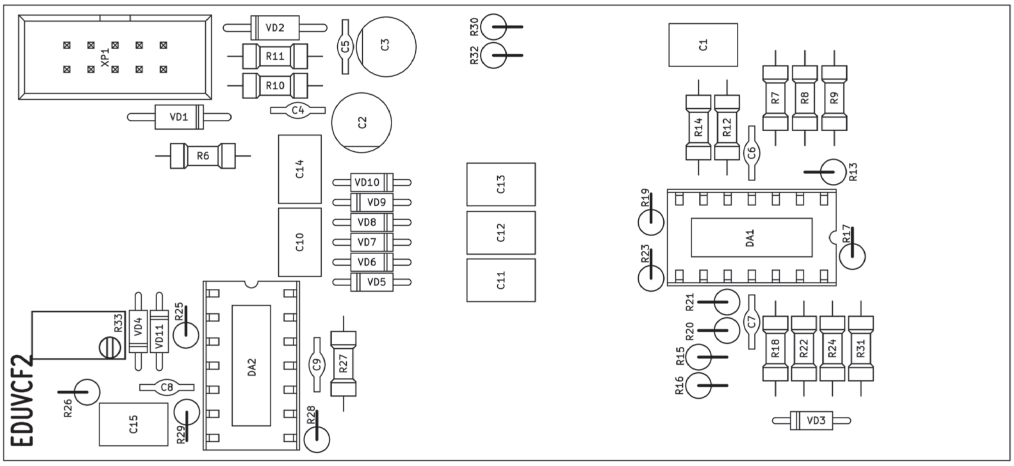
Capacitors **C6 – C9** are additional decoupling capacitors. If you inspect the PCB, you'll see that these are placed as close to the power supply pins of the ICs as possible. For well-designed, larger PCBs you will find decoupling capacitors next to each IC. Like the others, their job is to simply compensate for any unwanted noise in the supply rails. If the input voltage drops, then these capacitors will be able to bridge the gap to keep the voltage at the IC stable. And vice-versa - if the voltage increases, then they'll be able to absorb the excess energy trying to flow through to the IC, which again keeps the voltage stable. Typically, 0.1 uF capacitors are used for this purpose.

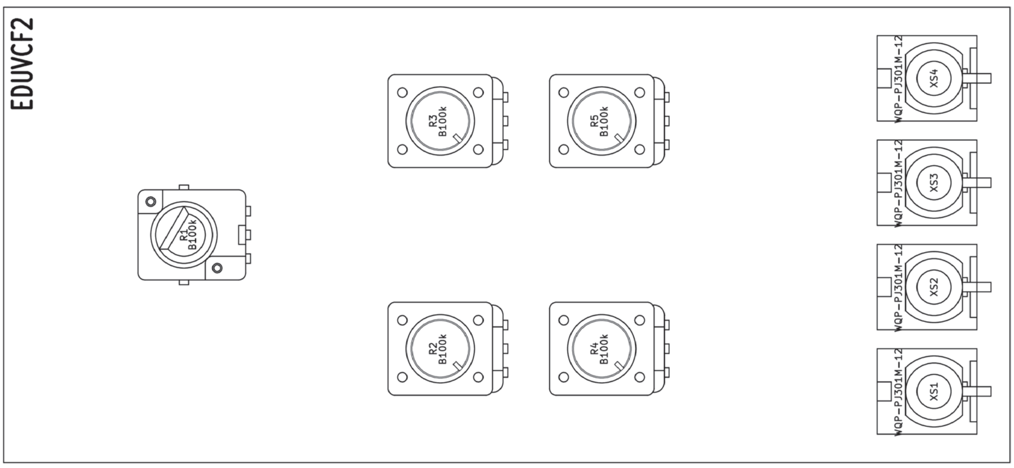
As mentioned in breadboard assembly part above, the best VCF performance will be with **matched diodes**. Contemporary components have good enough matching, so the VCF will work straight away, but, if you wish to try diode matching yourself, there's a kluge area for diode matching circuit on the right side of the PCB. Make sure, you have spare 1N4148 diodes for matching.

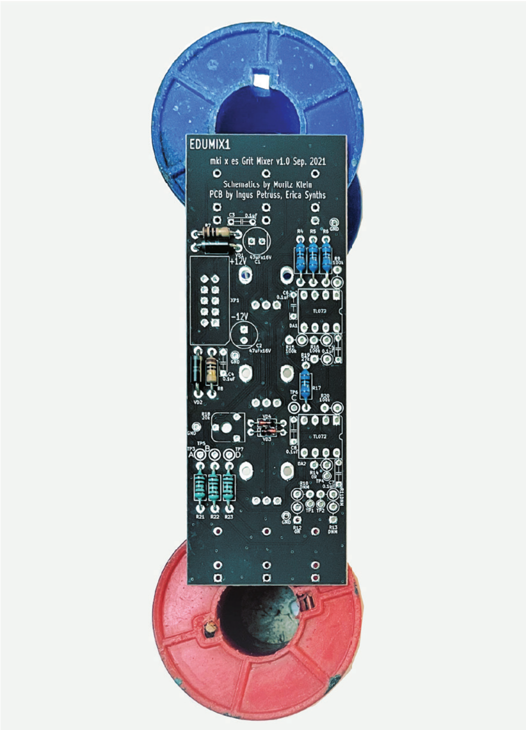

OPTIONAL DIODE MATCHING CIRCUIT
https://makingcircuits.com/blog/diode-matching-circuit-pairing-circuit/
https://www.fluke.com/en/learn/blog/digital-multimeters/how-to-test-diodes
RESONANCE INTENSITY
RESONANCE
OUT
IN
IN LVL
CUTOFF
CV1 LVL
CV2 LVL
EDUVCF2
mki x es
Sheet: /
File: EDUVCF2.sch
Title: DIODE LADDER FILTER
Size: A3
Date: 2021-10-25
KiCad E.D.A. eeschema 5.1.5
Rev: v2.0
Id: 1/1

Before you start soldering, we highly recommend printing out the following part placement diagrams with designators and values. Because some of our PCBs are rather densely populated, this will help you to avoid mistakes in the build process.

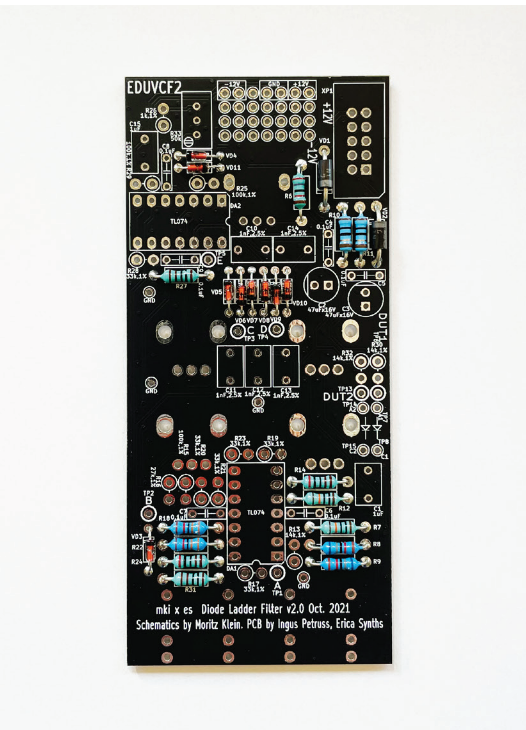




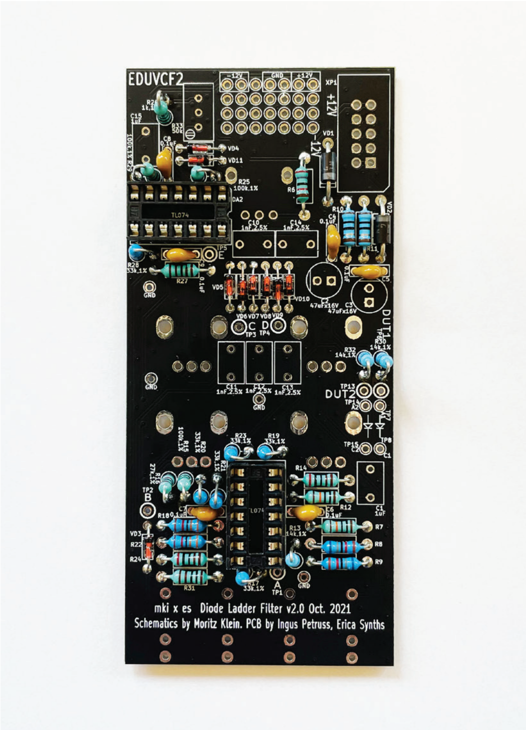




Place the VCF PCB in a PCB holder for soldering or simply on top of some spacers (I use two empty solder wire coils here).

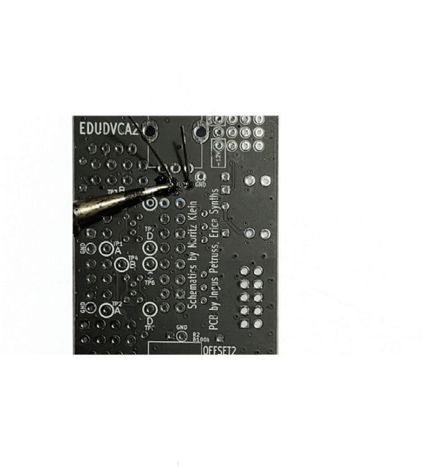


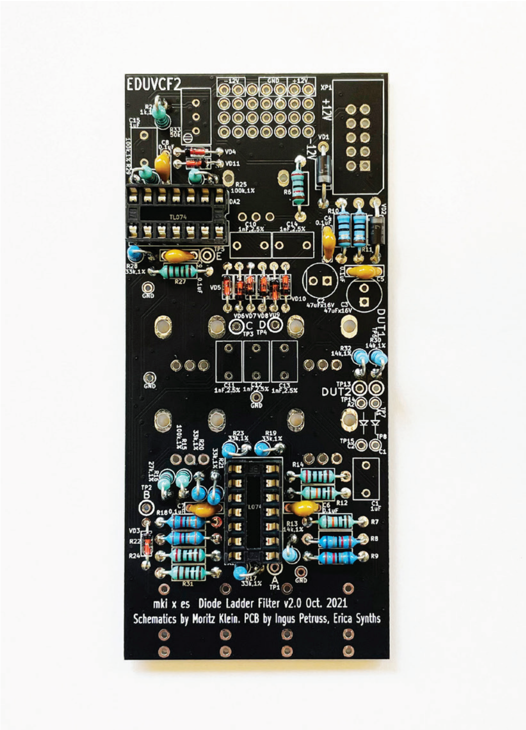
I usually start populating PCBs with lower, horizontally placed components. In this case, these are **some resistors, switching diodes** and the **power protection diodes.** Bend the resistor leads and insert them in the relevant places according to the part placement diagram above. All components on the PCB have both their value and denomination printed onto the silkscreen. If you are not sure about a resistor's value, use a multimeter to double-check. Next, insert the diodes. Remember – **when inserting the diodes, orientation is critical!** A thick white stripe on the PCB indicates the cathode of a diode – match it with the stripe on the component. Flip the PCB over and solder all components. Then, use pliers to cut off the excess leads.



Next, insert the first DIP socket, hold it in place and solder one of the pins. Continue with the **next DIP socket.** Make sure the DIP sockets **are oriented correctly** – the notch on the socket should match the notch on the PCB's silkscreen. Now, turn the PCB around and solder all remaining pins of the DIP sockets. Then proceed with the **ceramic capacitors.** Place the PCB in your PCB holder or on spacers, insert the capacitors and solder them like you did with the resistors & diodes before. Now your PCB should look like this:

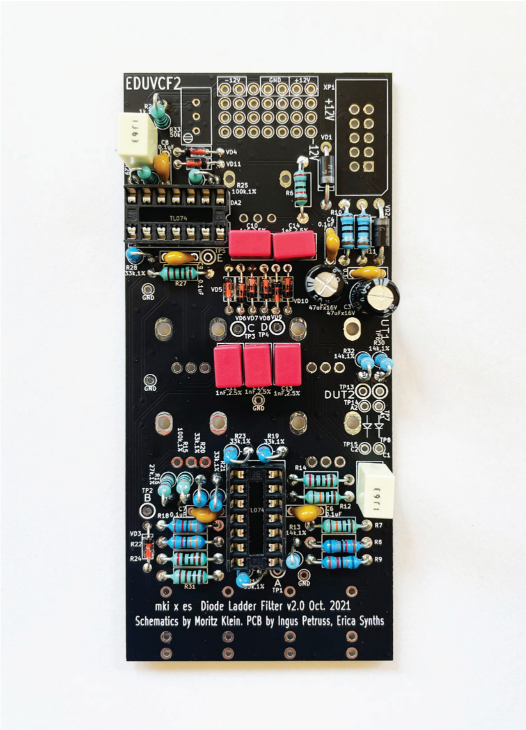
In order to save space on the PCB, some of our projects, including the VCF, have **vertically placed resistors.** The next step is to place & solder those. Bend a resistor's legs so that its body is aligned with both legs and insert it in its designated spot. Then solder the longer lead from the top side of the PCB to secure it in place, turn the PCB around and solder the other lead from the bottom. You can insert several resistors at once. Once done with soldering, use pliers to cut off excess leads.



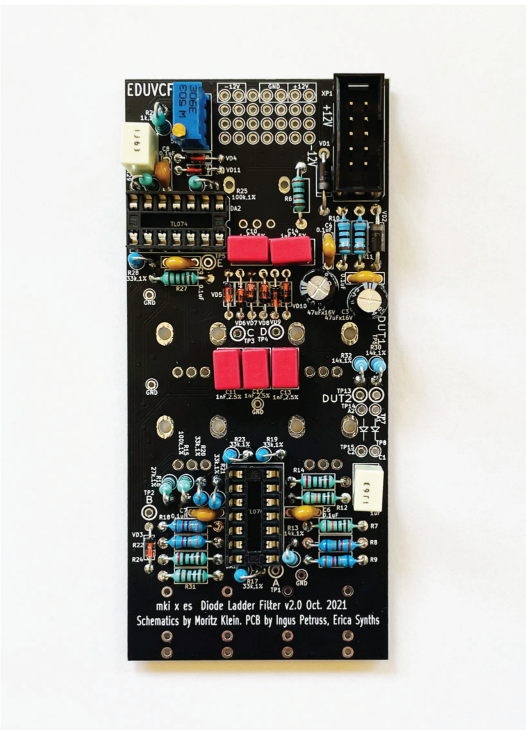


Once you are done with soldering all resistors, your PCB should look like this:

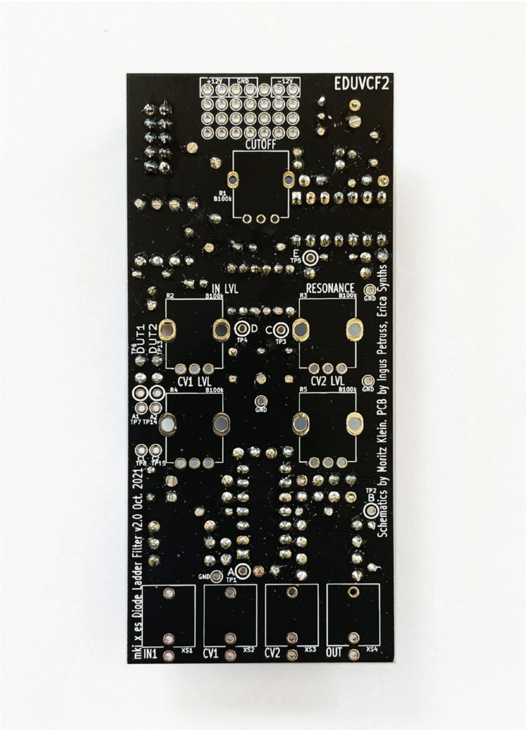
Next up: insert & solder the electrolytic capacitors. Electrolytic capacitors are bipolar, and you need to mind their orientation. The positive lead of each electrolytic capacitor is longer, and there is a minus stripe on the side of the capacitor's body to indicate the negative lead. On our PCBs, the **positive pad for the capacitor has a square shape,** and the negative lead should go into the pad next to the notch on the silkscreen.



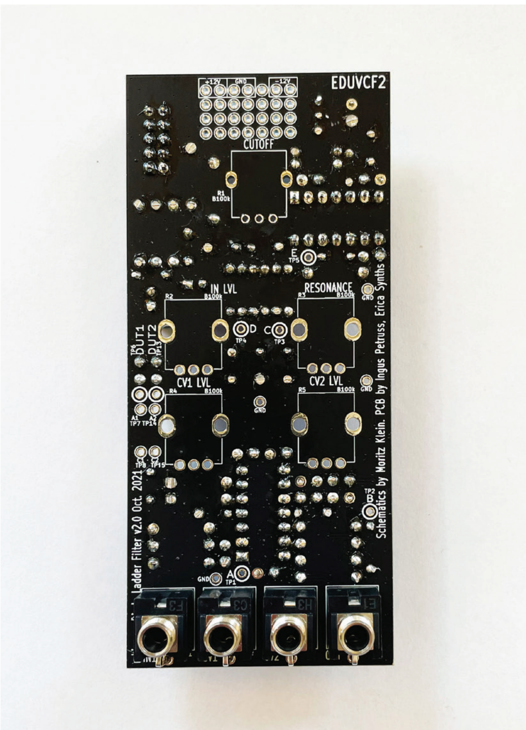
Then solder **film capacitors.** They are not polarized and can be soldered in any orientation. Here's how you PCB should look now.



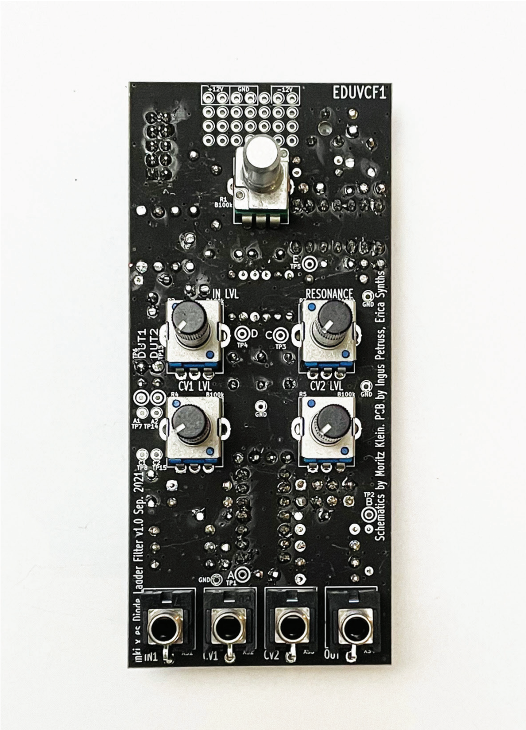
Then complete the component side of the VCF PCB by soldering the **PSU socket and multiturn trimmer potentiometer.** Make sure the orientation of the socket is as shown in the picture below – the arrow pointing to the first pin is aligned with a notch on the silkscreen. The key on the socket will be facing outwards the PCB. Now your PCB should look like this:



Now, turn the PCB around and inspect your solder joints. **Make sure all components are soldered properly and there are no cold solder joints or accidental shorts.** Clean the PCB to remove extra flux, if necessary.

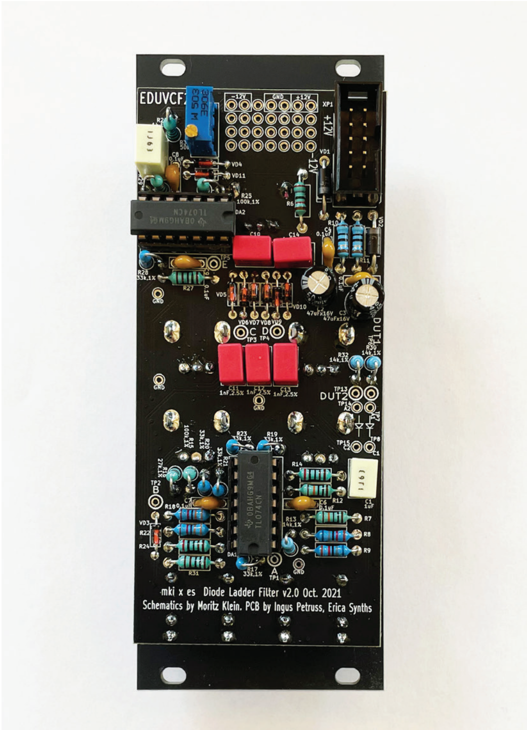


Insert the jack sockets and solder them.



Insert the potentiometers, but don't solder them yet! Fit the front panel and make sure that the potentiometer shafts are aligned with the holes in the panel – and that they're able to rotate freely. Now, go ahead and solder the potentiometers. Please note, that a green potentiometer with a metal shaft goes in a top middle position.

Install the **front panel** and fix it in place with the 5 hex nuts on the jack sockets and a washer + hex nut on the top potentiometer. Fit the large knob on the CUTOFF potentiometer and fix it in place with a small screwdriver. We are almost done!



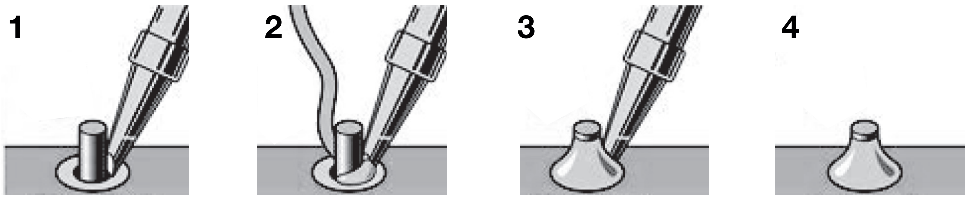
Now, **insert the ICs into their respective DIP sockets.** Mind the orientation of the ICs – match the notch on each IC with the one on its socket.

Congratulations! **You have completed the assembly of the mki x es.edu VCF module!** Connect it to your eu rorack power supply and switch it on. If there’s no "magic smoke”, it’s a good sign that your build was successful. Patch audio signal (your DIY.EDU VCO will be the best choice) to the input of the module and connect the output of the module to a mixer. Tweak the cutoff knob and check, if you have timbe change. Now you can proceed with calibration, which consists of adjusting the multiturn trimpot for desired resonance response. Please, follow instructions above! **Enjoy!**

SOLDERING APPENDIX

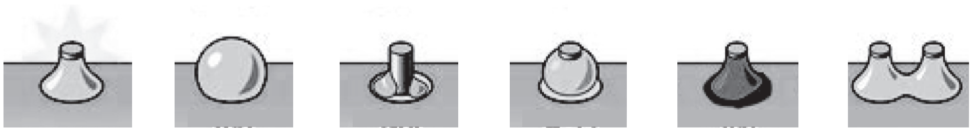
If you've never soldered before – or if your skills have become rusty – it's probably wise to check out some **THT** (through-hole technology) **soldering tutorials on YouTube**. The main thing you have to remember while soldering is that melted solder will flow towards higher temperature areas. So you need to make sure you apply equal heat to the component you are soldering and the solder pad on the PCB. The pad will typically absorb more heat (especially ground-connected pads which have more thermal mass), so keep your soldering iron closer to the pad on the PCB. It's critically important to dial in the right temperature on your soldering station. I found that about 320 °C is the optimal temperature for most of parts, while for larger elements like potentiometers and sockets, you may want to increase that temperature to **370 °C.**

Here's the recommended soldering sequence:



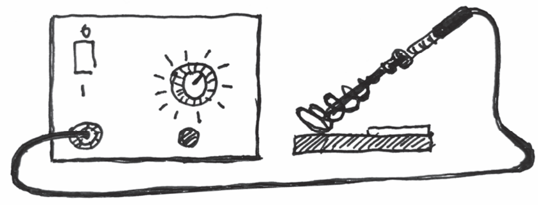
Heat part and pad 2 - 3 sec	Add solder	Continue heating 1 -2 sec.	Let cool

After you have completed soldering, inspect the solder joint:

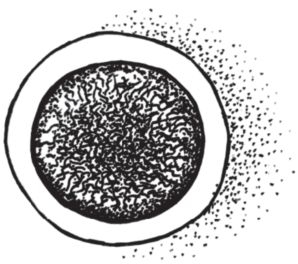


Perfect	Too much solder	Not enough solder	Cold joint	Too much heat	Short

DIY electronics is a great (and quite addictive) hobby, therefore we highly recommend you invest in good tools. In order to really enjoy soldering, you'll need:

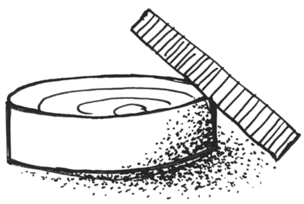


A decent soldering station. Top-of-the-line soldering stations (brands like Weller) will cost 200€ and above, but cheaper alternatives around 50€ are often good enough. Make sure your soldering station of choice comes with multiple differently-sized soldering iron tips. The most useful ones for DIY electronics are flat, 2mm wide tips.



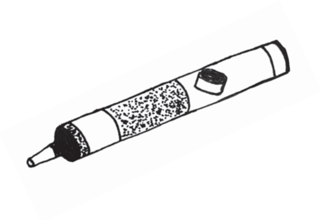
When heated up, the tips of soldering irons tend to oxidize. As a result, solder won't stick to them, so you'll need to clean your tip frequently. Most soldering stations come with a **damp sponge for cleaning the iron tips** – but there are also professional solder tip cleaners with **golden curls** (not really gold, so not as expensive as it sounds). These work much better because they do not cool down the iron.

Solder wire with flux. I find 0,7mm solder wire works best for DIY projects.

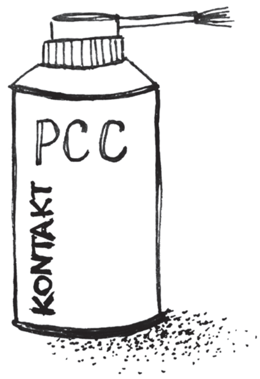


Some **soldering flux** paste or pen will be useful as well.

Cutting pliers. Use them to cut off excess component leads after soldering.



A solder suction pump. No matter how refined your soldering skills are, you will make mistakes. So when you'll inevitably need to de-solder components, you will also need to remove any remaining solder from the solder pads in order to insert new components.



Once you have finished soldering your PCB, it's recommended to remove excess flux from the solder joints. **A PCB cleaner** is the best way to go.

All of these tools can be found on major electronic components retailer websites, like Mouser, Farnell and at your local electronics shops. As you work your way towards more and more advanced projects, you'll need to expand your skillset and your tool belt – but the gratification will be much greater.

"Music is the strongest form of magic."

– Marilyn Manson