

JLCPCB provides an ordering [HOWTO](#). This may be all you need. What follows here is only something I started to try to provide the same instructions. I suggest starting with JLCPCB's guidance which may be more up-to-date and then, if it helps at all, read mine as well. You may not need to...

Make an Account and Begin Order

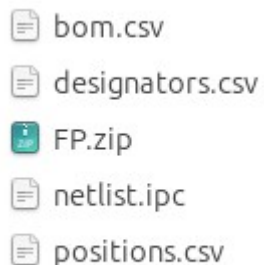
You will need to make an account for yourself on the [JLCPCB web site](#). Once you've done that and logged in, select "Order Now". The page that comes up will expect you to drop a **.zip** compressed file containing several Gerber files. This single .zip file provides all the information necessary for JLCPCB to create a bare PCB.

Generally after ordering, you'll get several emails during the next few days along the way as they fabricate the PCB and, if you also ask for PCB assembly (PCA), those production processes as well. You'll get a chance to look with 2D and 3D viewers etc.

Specify the Bare PCB

Using the [Field Probe](#) project as an example of how to use JLCPCB to obtain a finished and assembled PCB, first download the zipped file found in the "[Download Field Probe Kit](#)" in the material list at the bottom of its page. Unzip that file into a local directory of your choosing.

You should find something like the following example for the Field Probe but for the kit you want to build:



In this case, the FP.zip file contains all the Gerber files necessary to make a raw PCB. You don't need to unzip this, just drop it onto the waiting JLCPCB page.

The JLCPCB site should accept that .zip file and expect you to proceed with particulars for that raw PCB, e.g. material, how many you want, thickness - the defaults are often all OK, minimum quantity will be 5 etc. You have the option to ask them confirm the production file which will cause a short delay and an extra email exchange but may be desirable if there is any doubt about the design.

You can probably accept defaults, though you can change them if you like. Later, once you've actually placed your order if you've checked "confirm..." you'll probably get some extra email in a day or two asking for you to check what they plan on fabricating and want a confirmation that it's OK for them to proceed to production.

There will be running statement of Charge Details on the right as you proceed. This will be updated as you make selections.

At the bottom of that page there is a slider to select in order to continue to PC Assembly. You will select this and continue to PCA unless you are only ordering bare PCBs.

Specify the Assembly, PCA

After filling out and accepting the bare PCB description, at the bottom of the page, there is a slider to proceed to PCA if you wish. This will become a separate order in addition to the bare PCB. Some of the bare boards you have specified in the preceding step will then have parts placed and soldered.

As for bare PCB you will first select details including quantity and whether you want them to add tooling holes for you. Unless the design already has them it's generally fine to accept the default which is to have them added by JLCPCB. Confirmation of parts placement is also selected here. Don't worry if it looks wrong for some parts as that will be repaired later by JLCPCB.

Once you've accepted or changed the defaults, select "NEXT" on the bottom right of the Charge Details column. This will bring up a 2D representation of the PCB, assembly type and quantity. If it's what you expected, select NEXT on the right. Minimum assembly quantity if you enter it) will be 2. Any left over bare PCBs from the first step above will be shipped to you along with the assembled ones.

The next page will have you drag&drop from your local unzipped directory the **bom.csv** file, which describes the components and the **positions.csv** files which specifies assembly - where and how they will be placed on the raw PCB and soldered.

After dragging and dropping these files from your local directory has processed successfully (look for any warnings/complaints) select "Process BOM & CPL" to proceed. You will come to a page that shows the Selected (or not!) parts list.

At this point you may get an Error window saying that not all parts will be assembled due to data missing and asking if you want to re-upload the file or continue. As long as this lists parts that aren't actually assembled, sometimes details of the PCB that it thinks are parts but aren't, or that you don't want to have assembled (perhaps because they don't even exist in the .bom file) you can select Continue to override the warning. Otherwise you'll need to go back and resolve the unassembled parts. Selecting Continue is usually a safe response.

The next step is important and will probably take most the effort required in this entire process.

JLCPCB can only assemble and solder parts with C#'s they know about and have available parts.

The N6GN designs attempt to use Basic In Stock parts so that all of the parts will normally show up as 'Selected' in blue to the right of the page offered. However, for any PCA project, component availability changes over time so any missing, low quantity or other problems will show up at the end/bottom in red and must be resolved before continuing if you want them to be assembled. Some parts may not be in sufficient quantity and show "Shortfall" or even be "unavailable". You may decide

it's OK not to have them assembled if you don't need them or if you plan to complete assembly yourself when you receive the order.

If you do actually want them assembled by JLCPCB you may now select alternate parts using the magnifying glass. To change the request to a different C# Click on magnifying glass next to shortfall part, erase the old part name or number from the search box that pops up, fill in the Cxxxxxx number for the new part and then when the description appears click on Select.

Changing the C# and part amounts to a redesign so the onus is then on the kitter to accept responsibility for the changes. Downloaded 'binary' designs that don't have strikethrough have already been fabricated and are believed to work as described. Changing to a different part is certainly acceptable but the determination must move to the kitter who is changing it.

You resolve any "Unselected" parts either by

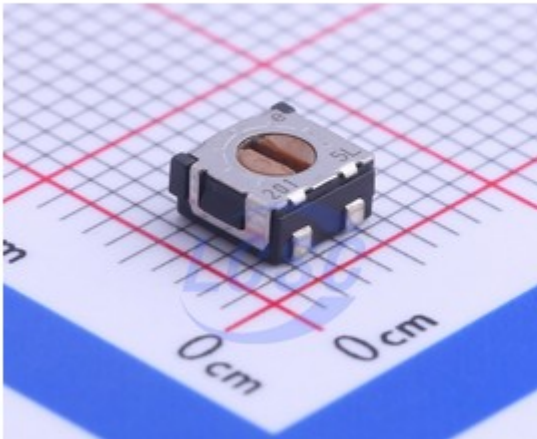
- searching for and selecting a substitute
- having JLCPCB pre-order them and place them in MyPartsLib (you pay up front)
- or by getting them from the Global Market (you pay up front for this too).
- In the worst case where no part or substitute is available worldwide you may have to use the (provided) Source Code to redesign using parts that are currently available.

You either need to OK the assembly with these parts NOT assembled, meaning you'll get a finished product that you have to finish assembly yourself after you receive the mostly-assembled PCBs or you'll need to go to the Parts page and have them pre-ordered. This will cause you to have to wait until they arrive at JLCPCB and become ready by being placed in your "MyPartsLibrary" for you to use. If you choose this, you will receive mail when they are ready to be used. This could take a day or two for parts JLCPCB supplies and perhaps 10 days for parts you specify and order through "Global Sourcing". If JLCPCB can supply them there will probably be extra fee per part number. Often they get them from a sister company, [LCSC](#) which may be nearby and who has many more parts than JLCPCB "Basic" parts.

LCSC has a much larger inventory of parts than does JLCPCB. Using that site and its search tool you may be able to find a likely replacement C# to specify for which there is stock on hand.

Here's an example of finding a substitute part via LCSC's site and search engine.

In this case I put in originally specified C132875 into LCSC search and backed out of the path shown



Nidec ST-4EA 200 ohm (201)







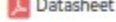

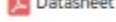

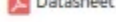

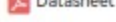

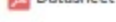
Manufacturer	Nidec
Mfr. Part #	ST-4EA 200 ohm (201)
LCSC Part #	C132875
Package	SMD

and clicked on "Potentiometers, Variable Resistors" to get only

there I selected 200 ohms, 1 turn , In Stock, and SMD with several package types that seemed reasonably close (and anything else I was sure that I needed but wasn't too exclusive, e.g. there might be multiple footprints that fit on the PCB pads OK.

With Smart Filtering turned on and while watching the "Remaining" quantities at the top until it got reasonable.

This gave six results

Images	Pricing 	Quantity	Availability 	Mfr.Part # 	Manufacturer
  Datasheet	1+ \$0.5842 10+ \$0.4774 30+ \$0.4241 100+ \$0.3841 More	<input type="text"/> Min: 1 Mult: 1 Full Reel: 1000	961 (0 + 961 + 0) In Stock	3313J-1-201E	BOURNS
  Datasheet	5+ \$0.1225 \$0.1280 50+ \$0.1020 \$0.1073 150+ \$0.0906 \$0.0953 500+ \$0.0837 \$0.0884 More	<input type="text"/> Min: 5 Mult: 5 Full Reel: 2500	592 (0 + 0 + 592) In Stock	TC33X-2-201E	BOURNS
  Datasheet	1+ \$0.8374 \$0.8814 10+ \$0.7548 \$0.7945 30+ \$0.7136 \$0.7511 100+ \$0.6826 \$0.7185 More	<input type="text"/> Min: 1 Mult: 1 Full Reel: 500	488 (0 + 226 + 262) In Stock	3314J-1-201E	BOURNS
  Datasheet	1+ \$1.0381 10+ \$0.9552 30+ \$0.9137 100+ \$0.8827 More	<input type="text"/> Min: 1 Mult: 1 Full Reel: 500	480 (0 + 480 + 0) In Stock	3314G-1-201E	BOURNS
  Datasheet	1+ \$0.5503 \$0.7146 10+ \$0.4608 \$0.5984 30+ \$0.4148 \$0.5387 100+ \$0.3701 \$0.4806 More	<input type="text"/> Min: 1 Mult: 1 Full Reel: 500	219 (0 + 11 + 208) In Stock	ST-4ETB201	Nidec
  Datasheet	1+ \$0.4052 10+ \$0.3958 30+ \$0.3911 100+ \$0.3848	<input type="text"/> Min: 1 Mult: 1 Full Reel: 500	3 (0 + 0 + 3) In Stock	ST-4ETA201	Nidec

Checking the highest availability parts with acceptable price and minimum quantity requirements I first tried **C124619** datasheet compared to the footprint on the board. Check this carefully to verify it will solder down OK. The exact pad being built on the raw PCB is described in the .zip of Gerber files and also in the Source PCB document. Sometimes the name of the package is all that is necessary but in any case Either a Gerber viewer or else KiCanvas can help discover those exact dimensions to verify that a potential substitute part's package match the spec sheet of a proposed part so can actually be soldered down.

For a cheaper option but with half the available stock **C720646** which is open frame but probably fits, again verify this.

Once a suitable available C# is identified, I then Select it using the microscope and pre-order that from JLCPCB. Usually in a day or two email will be received saying it is available for use.

If you've had to procure parts, you'll be notified once they arrive and you'll have to start the entire "Order Now" process again to be sure they are available or else use the BOM Tool to verify all necessary parts can be supplied. If you've managed to either specify a different part number or pre-order parts, those parts will no longer show up in red at the bottom but will be found just like all the others when you get to the step that didn't find them before. Parts you've had to had specially purchased either by way of JLCPCB or Global Sourcing will be place in MyPartsLib where they will automatically be discovered as available.

There will be a parts cost column showing what to expect each part number, extended for quantity used, to cost. This may be adjusted slightly later but usually it is correct.

Once all part selections have been resolved you may continue the PCA process. You may get a note saying that this has become a "Standard PCBA" order rather than an "Economic" one if some of the parts in the design require special attention. Normally you will just select to switch to Standard PCBA. You may get a reminder that not all parts have been selected if the previous step did not resolve every part on the .BOM. Just select "Do not place" if that is what is desired or else Select parts to go back and select them.

You may get a pop up reminder that components appear be offset from the PCB and an offer to automatically align them. Select OK.

Once you can get all the way through the PCA (PC assembly) step and show all the parts you want assembled in blue, they let you look at the 2D/3D rendering (not necessarily positioned the way it will actually come out) and finally they are ready for you to pay for the order. You don't have to continue here but this will show you what to expect if you do.

Note that they will show you a preview **approximation** of the assembled board with parts assembled. This may include errors in the way the parts seem to be positioned. Don't worry about this, it's mostly a difference between JLCPCB's positioning and KiCad's that will actually be fixed later after one more round of email after you've paid.

The parts will again be listed on the right column and after examining everything again, perhaps in 3D, you can again select NEXT.

You will now be presented with a final Charge Detail showing how much everything will cost, how long assembly will take, and on the right, an opportunity to SAVE TO CART. Prior to putting it in the cart they will ask for a Product Description (usage model for customs, I think) where selecting DIY seems appropriate.

Ordering and Paying

After getting comfortable that the PCB to be fabricated and parts to be assembled are what you want and expect you are ready to actually spend money and order what you've specified.

The final page will show the order(s) for bare PCB and PCB Assembly, quantities, build time and prices. There will be Secure Checkout and a chance to pay offered next.

For small quantities you will often end up paying at least double. Shipping, handling, setup .. and other one-time charges will make the total price much more than the sum of the raw PCB and the individual components (you will have seen the prices when you did selection and there will be a total price statement to look at too). A total cost for PCB, assembly, shipping, "engineering expense" and charges not related to the quantity will be available.

Because parts cost is very low for "Basic" RLC components, it often makes sense to order more assembled PCBs than you need, especially if others near you may like to have a completely assembled kit as well. The total cost will go up slightly but the per-piece cost will go down because the one time charges mentioned will be spread out among more boards.

You'll see a final bill and be offered several ways to ship and pay.

After ordering you may see adjustments to costs or fees and have receive a request for slight additional charges or provided with refunds. Either may occur but these are generally small amounts.