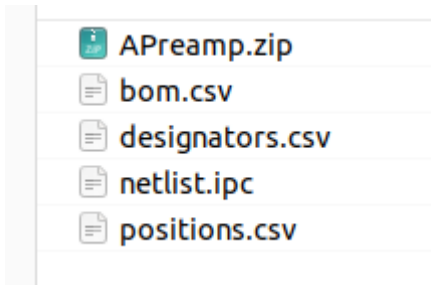As an example of how to use JLCPCB to obtain a finished and assembled PCB, here is an example kit package of "binary" code that can be dropped on [JLCPCB web site](#) .

You will need to make an account for yourself first, then select "Order Now" . Only three files in total need be used. First drop the **APreamp.zip** (Gerber) file in the top PCB fab area of the order page.

Below that you will verify and select the raw PCB quantity and characteristics - the defaults are probably all OK, minimum quantity will be 5.

Once that is complete you will proceed at the bottom to add the **bom.csv** file and the **positions.csv** files which specify assembly characteristics. Minimum assembly quantity (you must enter it) will be 2.

After it has processed this successfully (look for any warnings/complaints) you will come to a page that shows the Selected (or not!) parts list.

This is an important step. JLCPCB will only assemble/solder parts with C#'s they know about. Any missing, low quantity or other problems will show up for parts at the end/bottom and must be resolved before continuing.

You resolve them either by having them provide them (you pay up front) or by getting them from the Global Market (you pay up front for this too.

Once you can get all the way through this step 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, you may have found that not all the necessary parts are yet available or something but this will show you what to expect.

For small quantities you will 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).