

Download production and paste files

Download production and solder-paste files.

You can download the production files for all your MakeMyPCB orders directly from the website. You have immediate access to your original data and to all the CAM-verified files we have generated to produce and test the board. If your layout uses SMD components we automatically generate a solder-paste file which you can also download.

Proceed as follows:

1. Log in with your username and password.
2. Click on the Order tab on the left-hand side of the screen and then on the Repeat Order/View History icon for a completed order or on the Running Orders icon for a current order. For current orders you will only be able to download the data if the CAM work has been completed (status is no longer PREP).
3. Tick the order whose files you want to download.
4. Click the "DOWNLOAD PRODUCTION DATA" button at the top of the screen.
5. A pop-up will appear with the name of the archive file for your job. Click Download and the file will be downloaded to your PC.

To access the production files:

1. The file you have downloaded is called *SI-ordernr.zip*
2. Unzip this file. This will give you a file called *ordernr.dpf*
3. You may need to rename this file as *ordernr.zip*
4. Unzip again and you will get the production files including *ordernr-paste.zip*
5. Unzip *ordernr-paste.zip* and you will find the paste data in Gerber 274X and our internal CAM format DPF.

Paste files: a warning.

We recommend that you check the paste files before you use them to produce a stencil. We generate the paste files from the SMD pads on a "best-efforts" basis but it is not always clear to us which pads to include and which to leave out. It is much safer to generate a paste file yourself and include it in the data you send us. We will then check it with the rest of the data and if required step and repeat it into the same assembly panel.