

## Navodila za izdelavo Gerber in NCDrill datotek iz programa EAGLE 7

Veliko elektronikov me sprašuje, kako izdelati Gerber in Drill datoteke iz programa EAGLE za risanje shem in tiskanih vezij, tako, da mi je že kar nekoliko neprijetno, da izkušenj z Eaglom nimam. V podjetju uporabljamo Altium Designer, P-CAD 2006 in Target 3001. Za vse tiste, ki uporabljajo Eagle, pa sem raziskal, kako hitro in učinkovito pripraviti ustrezne Gerber in NCDrill datoteke, ter pri tem obdržati prav vse lase na glavi!

Zelo pomembno: za kreiranje datotek čisto brez vsakršnih nastavitev in težav potrebujemo CAM datoteko, ki jo dobimo na spletni strani spletne trgovine z elektroniko Sparkfun:

<https://www.sparkfun.com/tutorials/108>

**Beginning Embedded Electronics - 8**  
by Nate | June 19, 2008 | 67 comments  
Skill Level: ★ Beginner

### Lecture 8 - Eagle: Schematics

We are very proud of this tutorial but it is getting long in the teeth. Be sure to check out the original tutorial for your viewing pleasure:

- [How to install and setup Eagle](#)
- [Using EAGLE: Board Layout](#)
- [Using EAGLE: Schematic](#)

Here's the original tutorial for your viewing pleasure:

Welcome to the wonderful world of PCB creation! We've used a few software packages, **Layout Editor** from CadSoft to be very easy to use, very cost effective, and very powerful. Eagle is free! There are some limitations in place, but basic students and non-professionals can use it. Eagle is not the 'hobbyists' tool you may think it is. I've seen some very complex boards that were too was amazed to hear it was created in Eagle. It can be done, you just need to do it right.

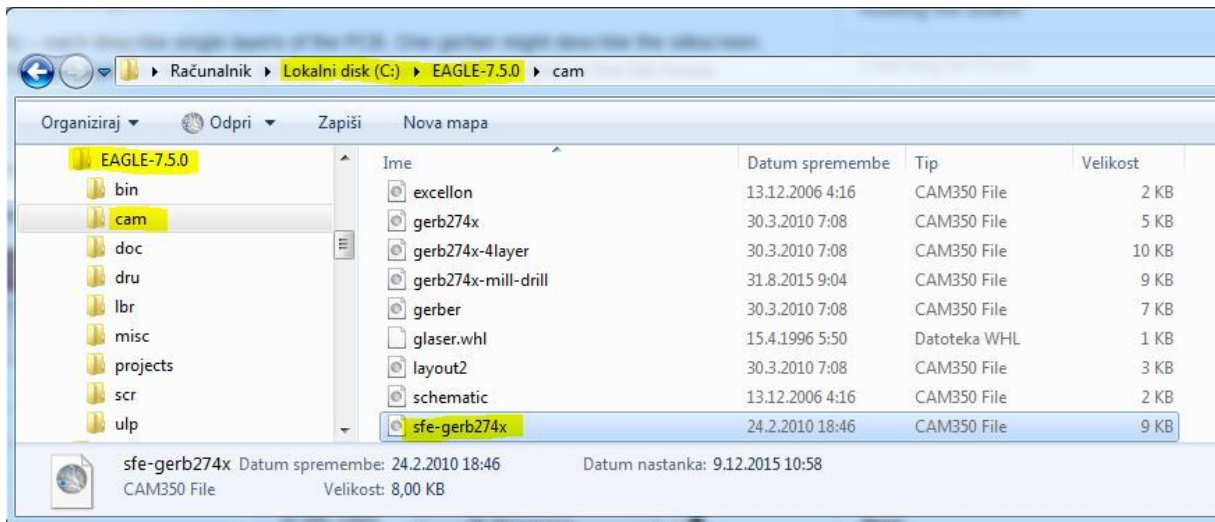
There are a few files that you will need to download for this workshop.

1. Download **Eagle** itself. Currently we use v4.16 (~8MB). Versions are available for Windows, Linux, and Mac. If the above link does not work, google 'eagle pcb download?' to get the latest version.
2. Download the **SparkFun Eagle Library**. This is the collection of all the components SparkFun designs with and therefore components and footprints that have been tested. Unzip and place the SparkFun.lbr file into the Eagle\lbr directory. If the above link does not work, google 'sparkfun eagle library?' to get the latest collection.
3. Download the SparkFun **Eagle keyboard shortcuts**. Place this file in the Eagle\scr directory. If the above link does not work, google 'sparkfun eagle shortcut?'.
4. Download the SparkFun **CAM file**. (Right-click, and choose Save Link As) Place this file in the Eagle\cam directory. This file is responsible for creating the gerber files for submission to a PCB fab house.

**Note:** The SparkFun Eagle shortcut key script file has an .scr extension. This is a common virus infiltration method. If you choose to download our keyboard shortcuts, and you don't trust us, rename the file to a .txt extension and view it in a text viewer. There's nothing there but text and Eagle commands. Just be sure to rename the file to the .scr extension so that Eagle will use it.

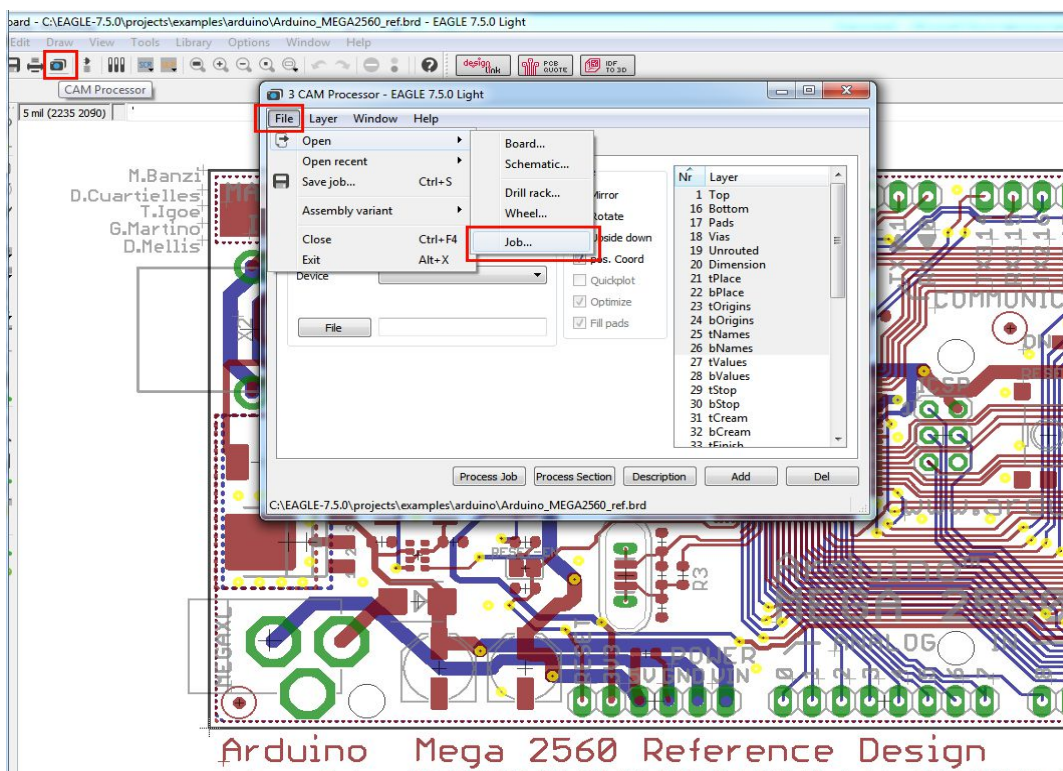
To learn how to use Eagle, we are going to create a simple breakout board for a popular USB IC. The FT232RL is a USB to TTL serial converter.

Izberemo možnost »Datoteko shrani« in jo shranimo v podmapo z imenom »CAM« vaše nameščene različice programa Eagle, pri meni je to na primer:

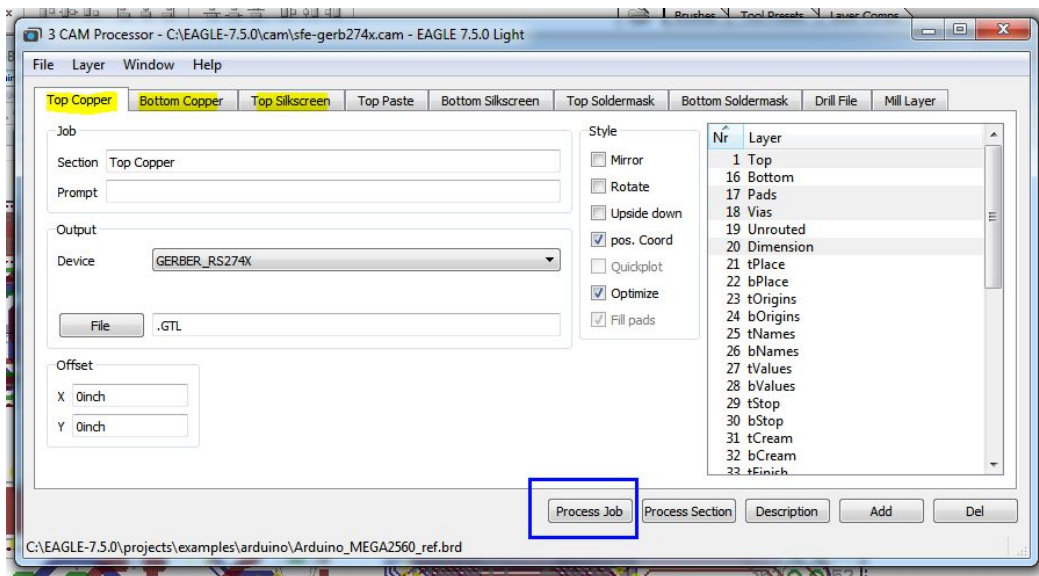


Datoteko bomo pri ustvarjanju Gerber in NCDrill datotek uporabili kot navodila za delo, »JOB«.

Ko želimo iz narisane ploščice ustvariti potrebne datoteke, izberemo ikono »CAM Processor«:



Ponudi se nam menu, izberemo prvo opcio, »Open« potem »Job«. Odpre se nam vsebina »CAM« mape, kjer najdemo tudi tisto CAM datoteko, ki smo jo prenesli s spletne strani Sparkfun. Izberemo prav to datoteko, ki za ustvarjanje vseh potrebnih datotek za izdelavo TIV vsebuje že vse potrebne nastavitve, končnice, plasti in poti za kreiranje datotek. Za vsako datoteko je poseben zavihek, kjer je nastavljeno vse, kar je potrebno za njeno ustvarjanje. Ne potrebujemo več ničesar drugega, le še gumb »Process Job« pritisnemo:



Če nadaljujemo kar z navodili, ki jih najdemo na Sparkfun:

The gerber generation process should be pretty quick. Once it's run its course, have a look in your project directory, which should have loads of new files in it. In addition to the board (BRD) and schematic (SCH) files, there should now be a .dri, .GBL, .GBO, .GBS, .GML, .gpi, .GTO, .GTP, .GTS, and a .TXT. Meet the Gerbers!

Gerber File	Extension
Bottom Copper	GBL
Bottom Silkscreen	GBO
Bottom Soldermask	GBS
Top Copper	GTL
Top Silkscreen	GTO
Top Soldermask	GTS
Drill File	TXT
Drill Station Info File	dri
Photoplotter Info File	gpi
Mill Layer	GML
Top Paste	GTP

Veliko uspeha vam želim!