

EXPORTING GERBER FILES FROM DESIGN SPARK

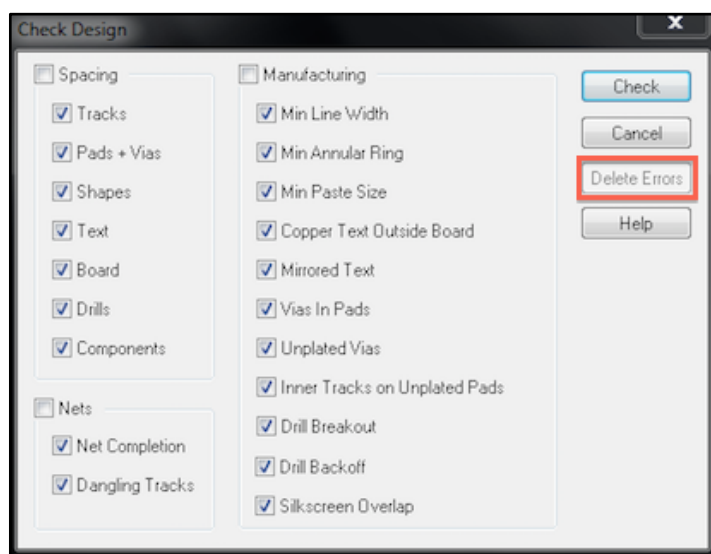
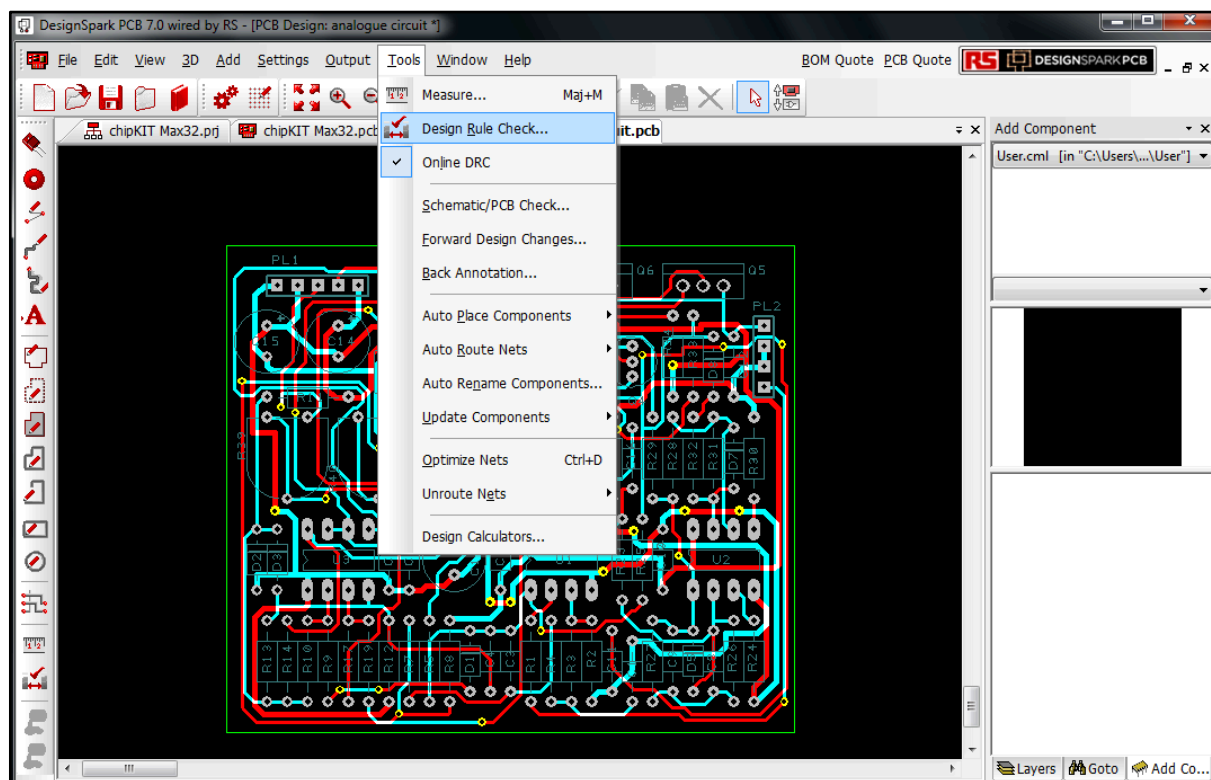
V1.1 – 09/2014

DESIGN RULES CHECK	1
GENERATING GERBER FILE	2
SENDING GERBER FILES	7

Design Rules Check

Before proceeding to Gerber and NC Drill files generation, a DRC (Design Rules Check) must be performed in order to check your layout respects all the manufacturing constraints.

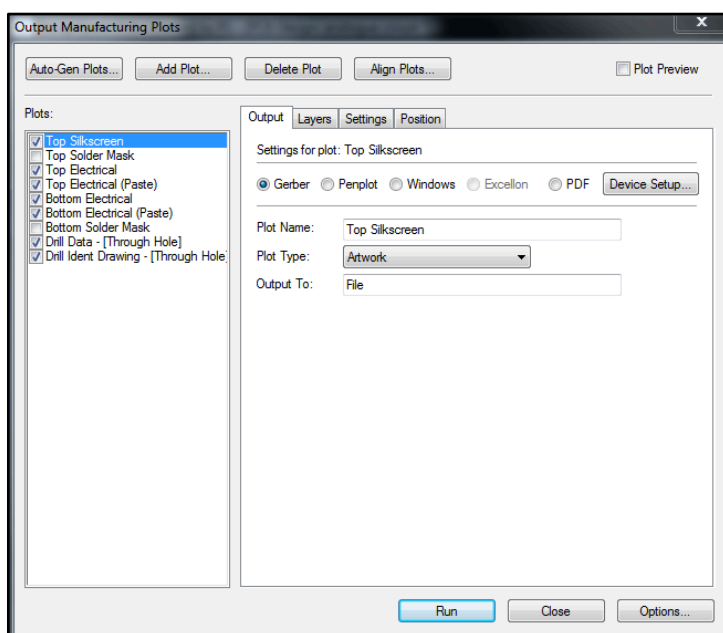
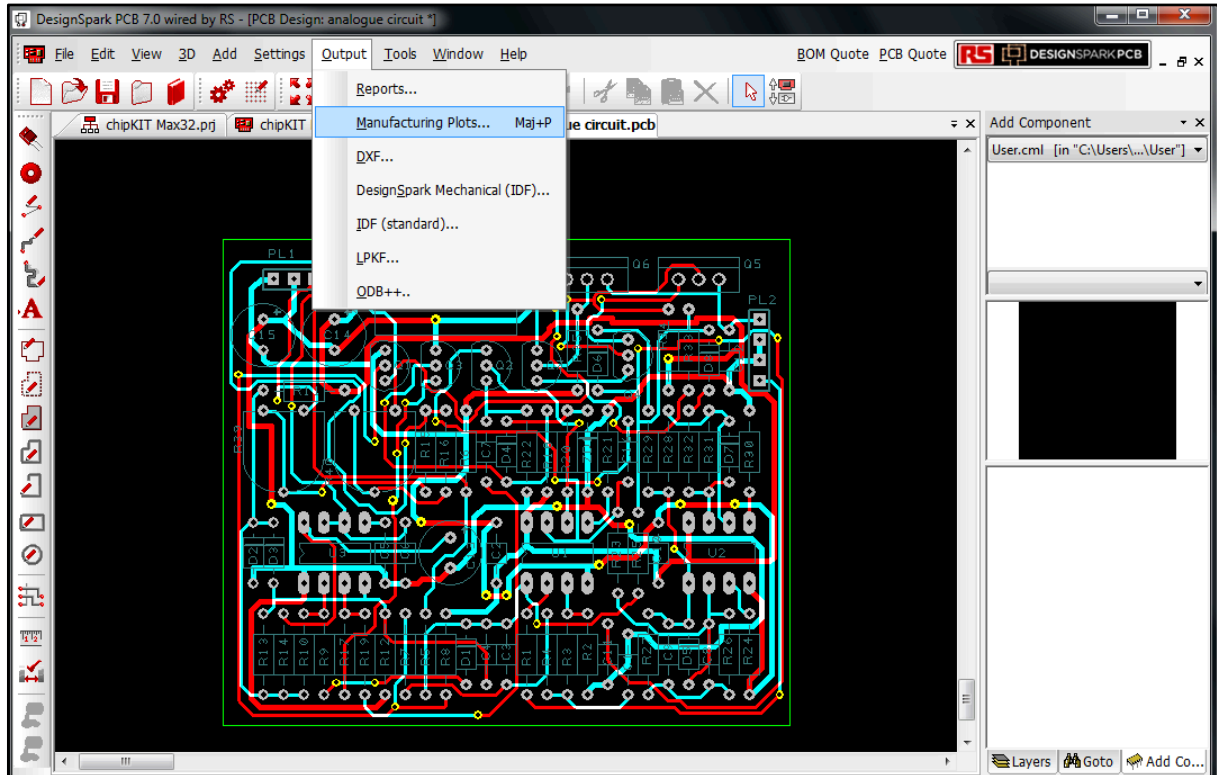
In Design Spark, select : **Tools / Design Rule Check** and run **Delete errors** in the "Check Design" window.



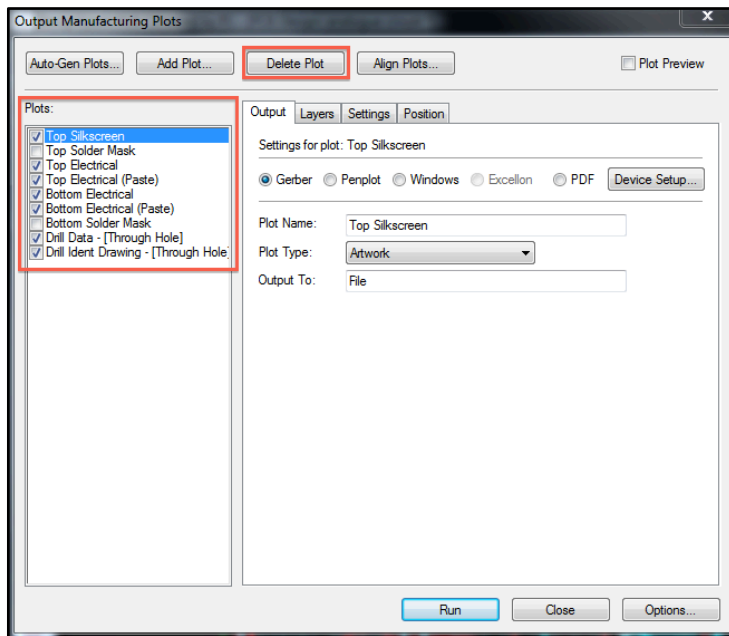
Correct all possible errors before proceeding to the next steps.

Generating Gerber File

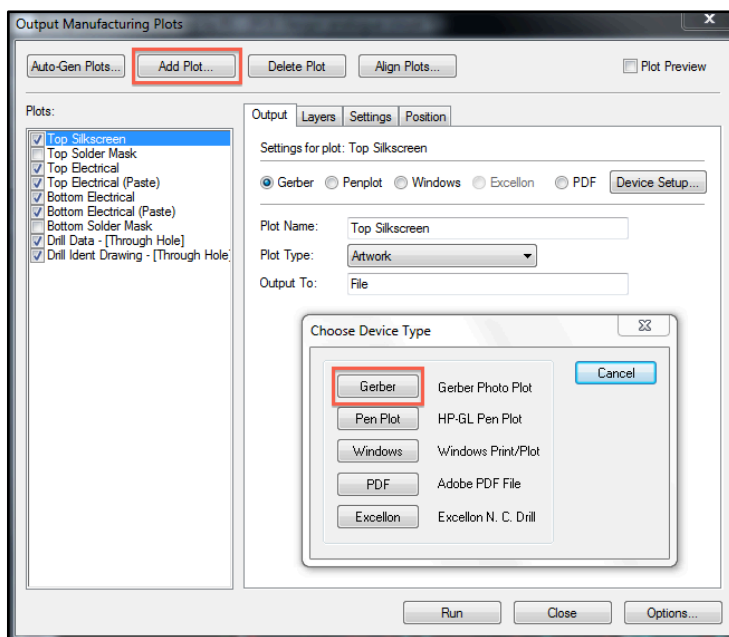
Click on [Output](#), then [Manufacturing Plots](#).



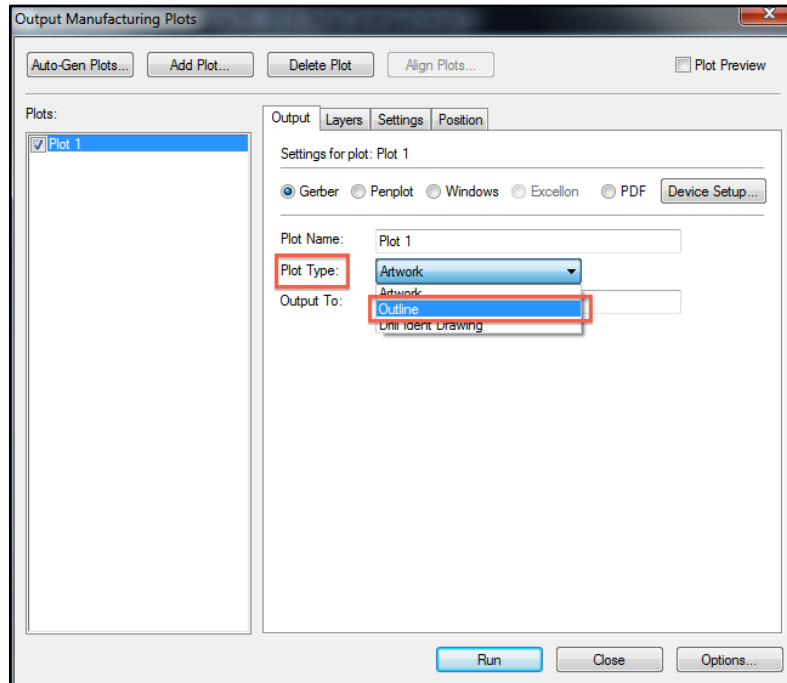
In the **Output manufacturing plots** window, delete the following plots (**Delete Plot** button):



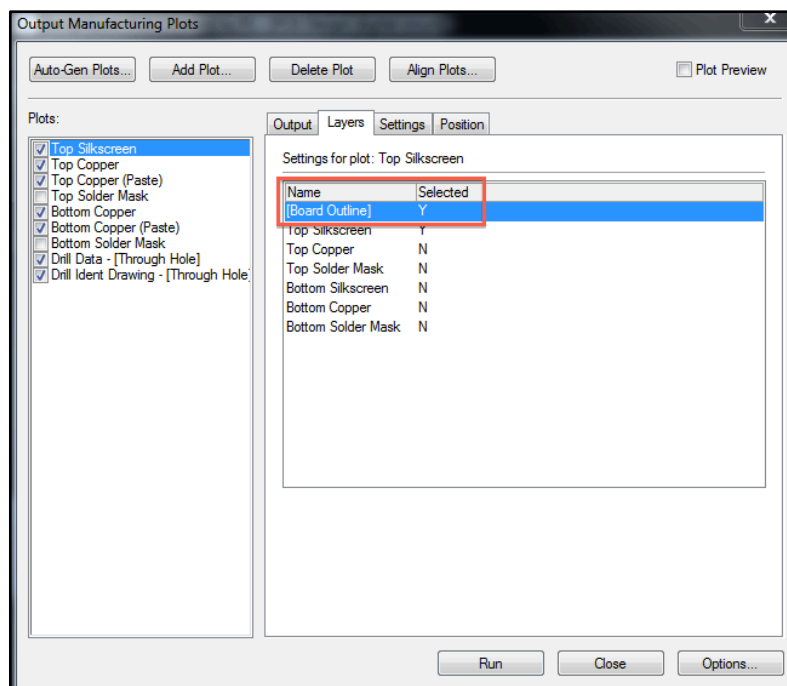
- **Top Copper (Paste)**, **Bottom Copper (Paste)** and **Drill Ident Data - [Through Hole]**.
- Uncheck all remaining boxes.
- Click '**Add Plot**' and select '**Gerber**' on the pop-up box.



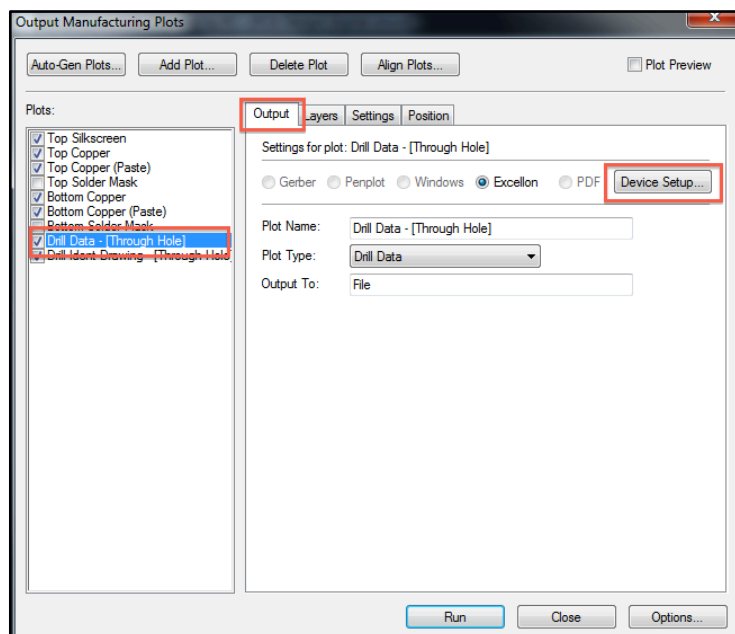
- Rename this Plot 1 into Board Outline, PAS TROUVÉ SUR LE SOFT
- Change the type to Outline.



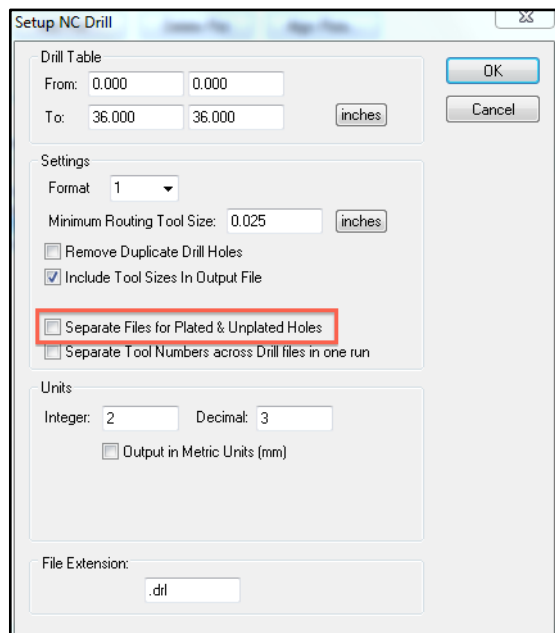
- Select the Layers tab and double click on Board Outline, to check "Y".



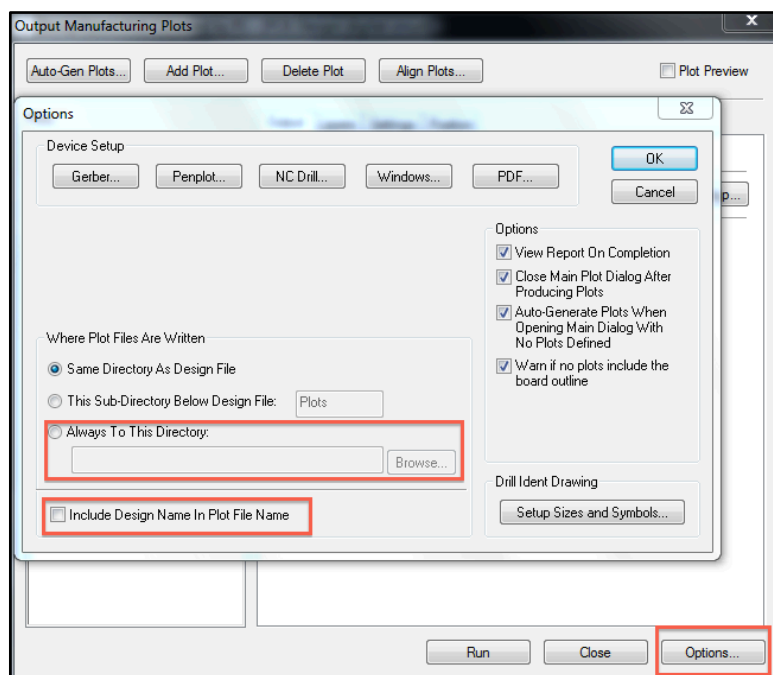
- Click on 'Drill Data - through Hole', and in the **Output** tab, click on the **Device Setup** button.



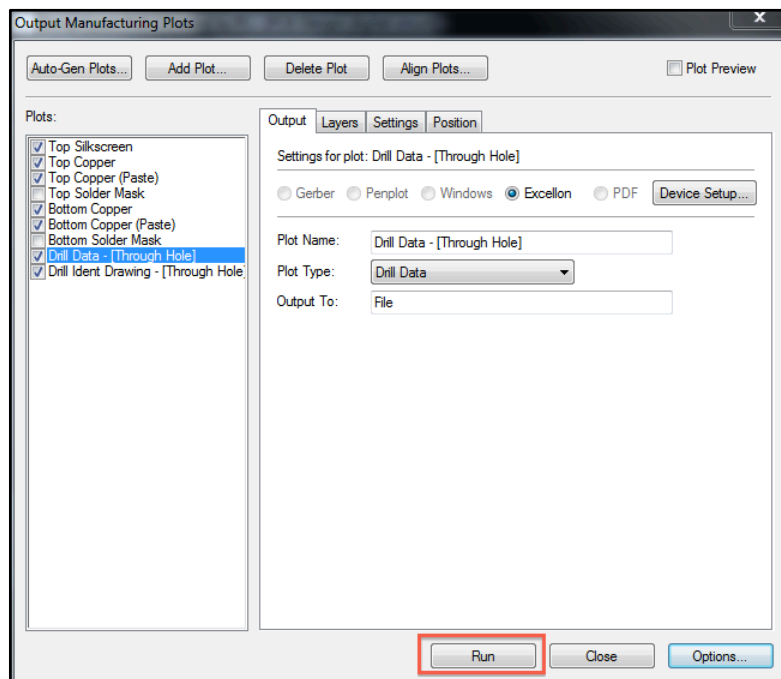
- Uncheck **Separate files for plated and unplated holes** if already checked. Then click **OK**.



- Click [Options](#) (bottom right) and uncheck [Include Design Name in Plot File Name](#). You can choose a destination file via [Always To This Directory](#).



Click [Run](#).



A Plot report (text file) should be displayed.

Sending Gerber Files

The file extensions need to be changed, so that E-circuits can read them properly. Please follow instruction in the following array :

File Extension		New extensions
Board Outline	.gbr	.gko
Bottom Copper	.gbr	.gbl
Bottom Solder Mask	.gbr	.gbs
Drill data - Through Hole	.drl	.xln
Top Copper	.gtr	.gbl
Top Silkscreen	.gbr	.gto
Top Solder Mask	ggbr	.gts

- Compress all the files in a single *.zip* file.
- Your files are ready to be sent.