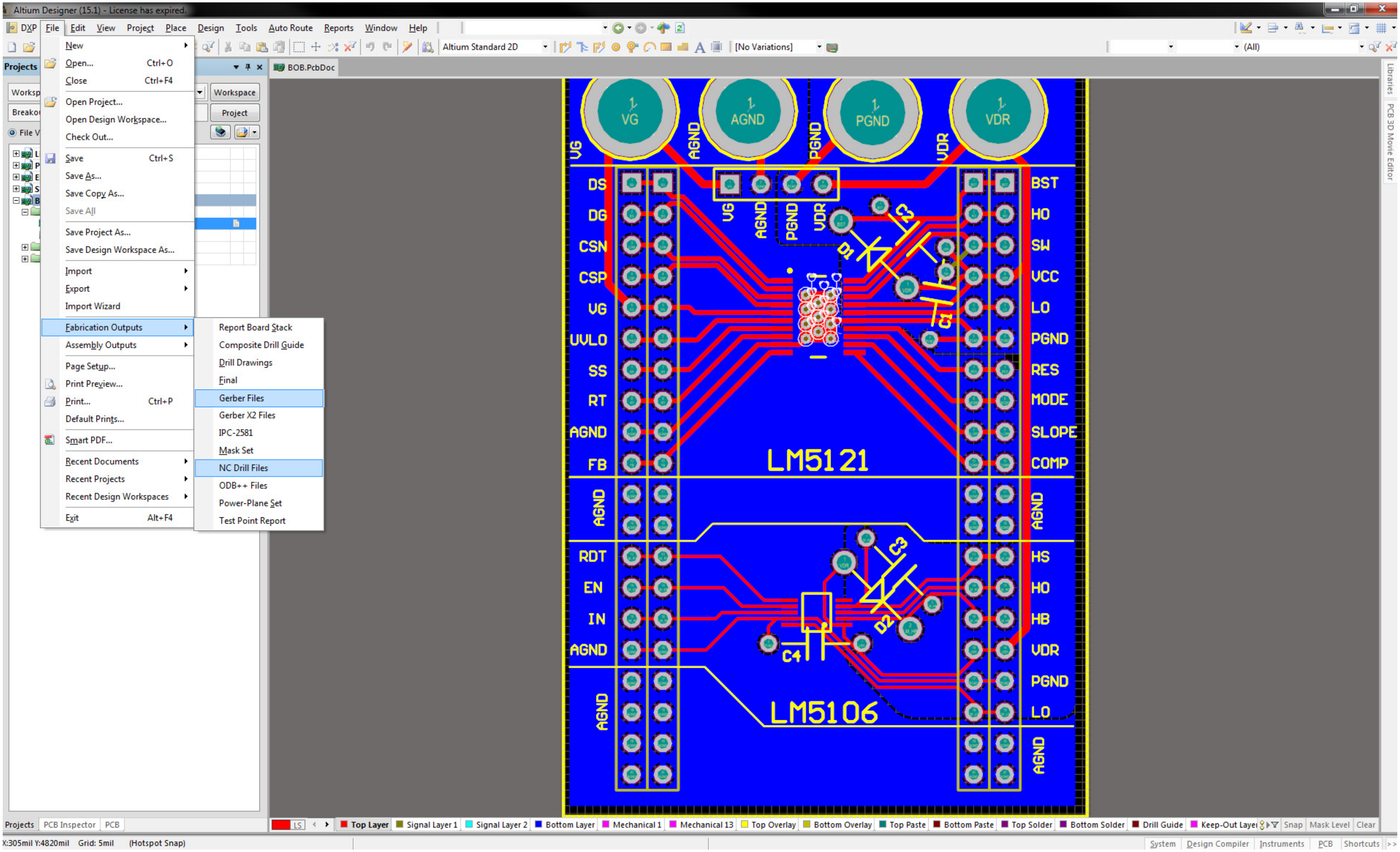
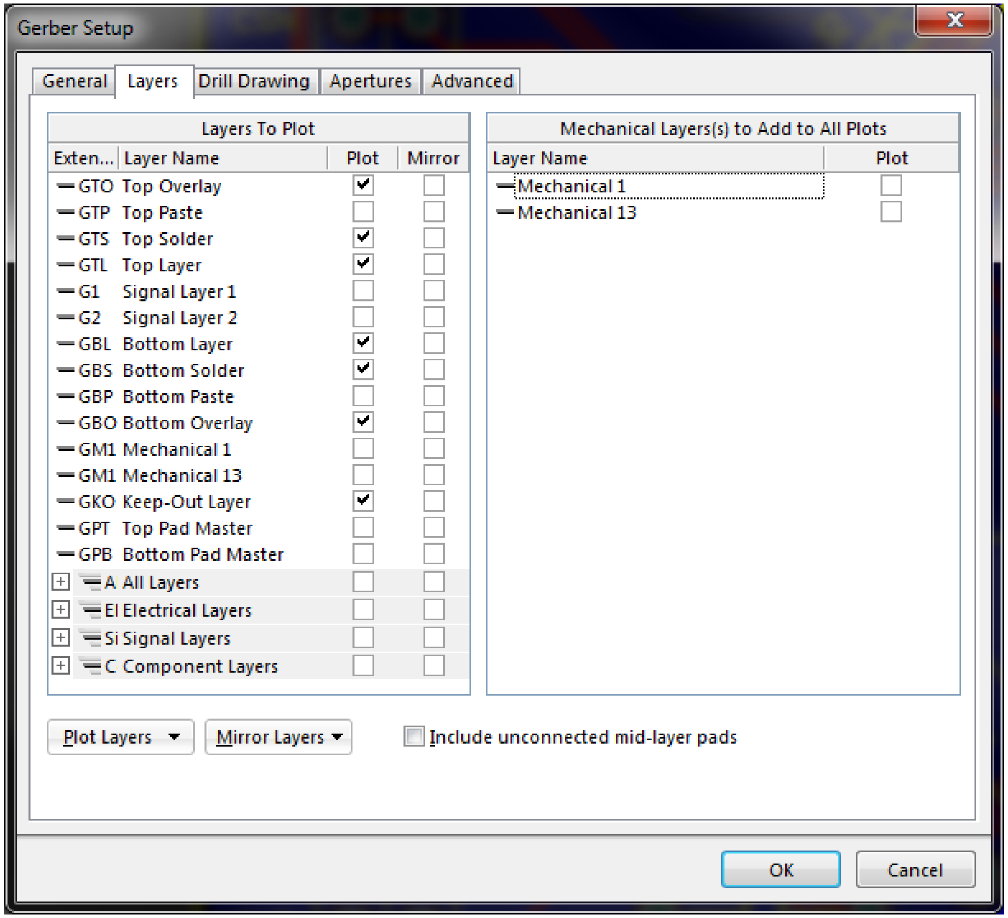
Submitting PCBs in Altium to Sierra Circuits

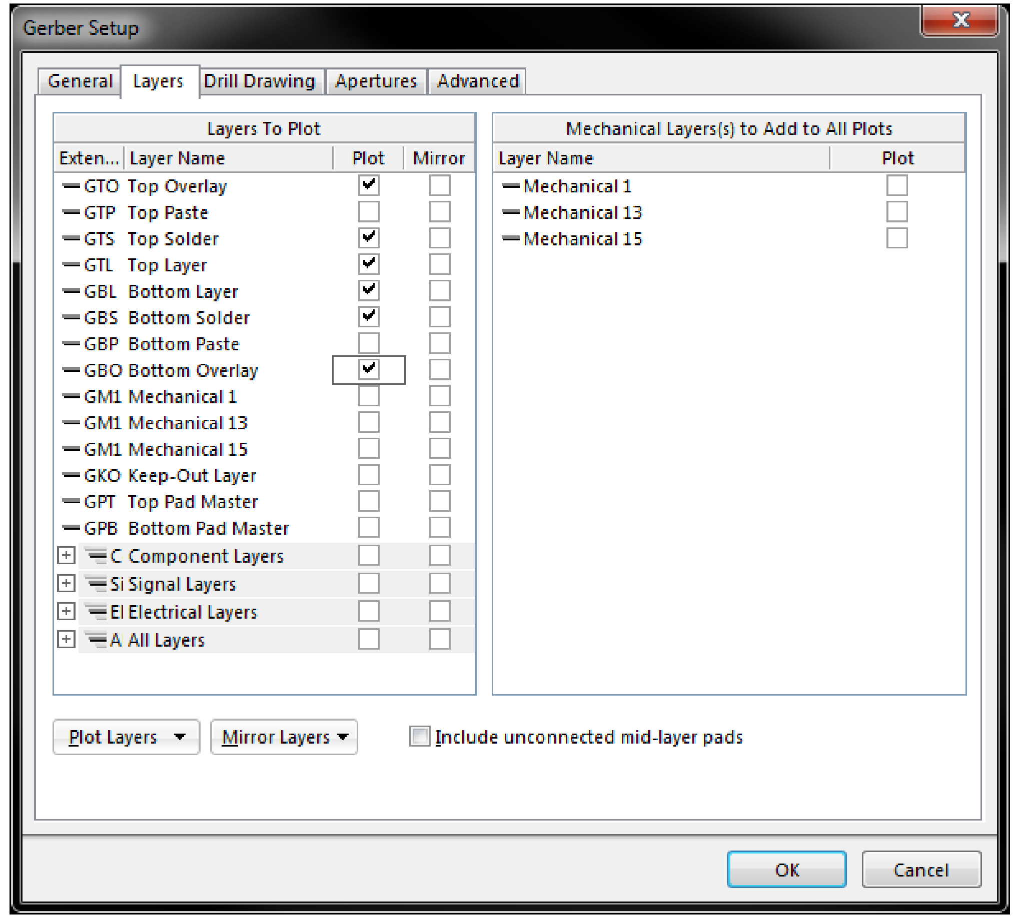


1. When your Layout is completed output both gerber files and NC Drill files

These are the CAD documents which will be used to tell the machines how to manufacture each layer of your board



4-layer Gerber Template



2-layer Gerber Template

2. In the gerber file dialog, Select the layers needed for the PCB.

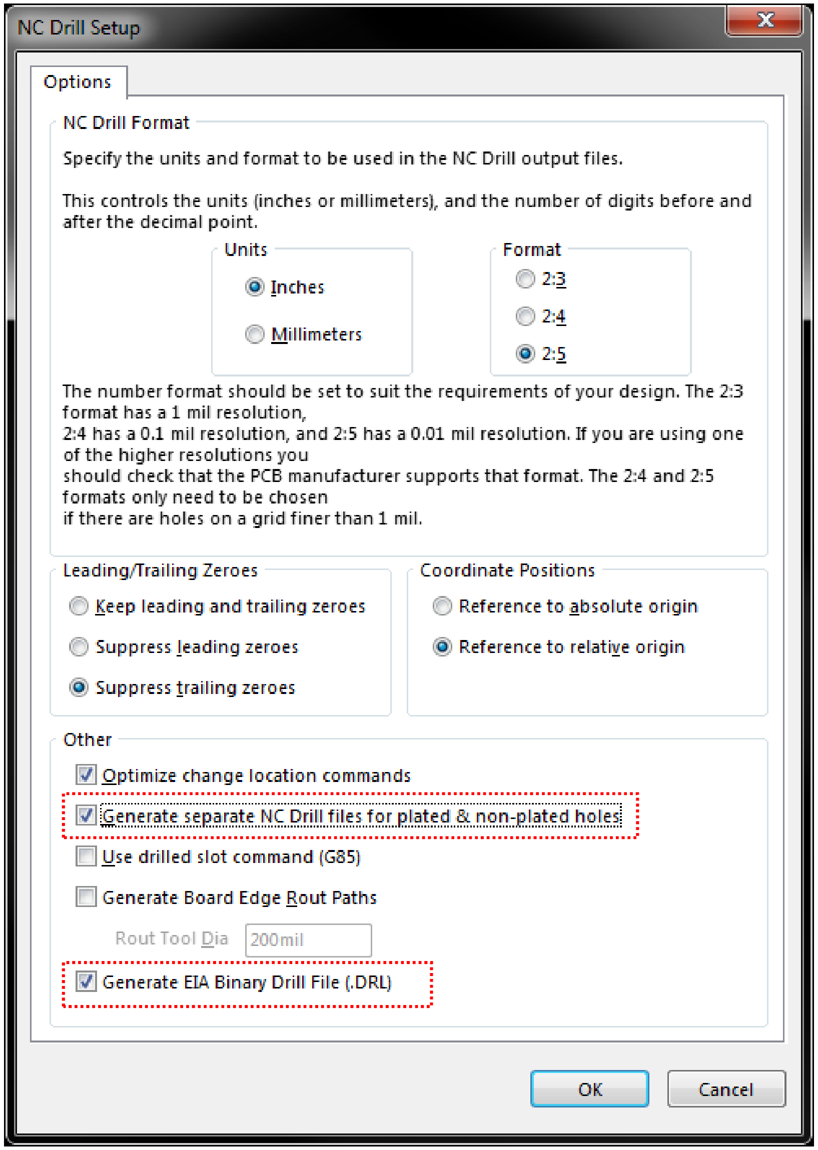
The necessary layers are:

1. GTL - Top Layer – Copper on the top of the board
2. GBL - Bottom Layer – Copper on the bottom of the board
3. GTO – Top Overlay – Silkscreen Layer on the top of the board
4. GBO – Bottom Overlay – Silkscreen Layer on the bottom of the board
5. GTS – Top Solder– Soldermask Layer on the top of the board
6. GBS – Bottom Solder– Soldermask Layer on the bottom of the board

   layer defines where soldermask *is not* present

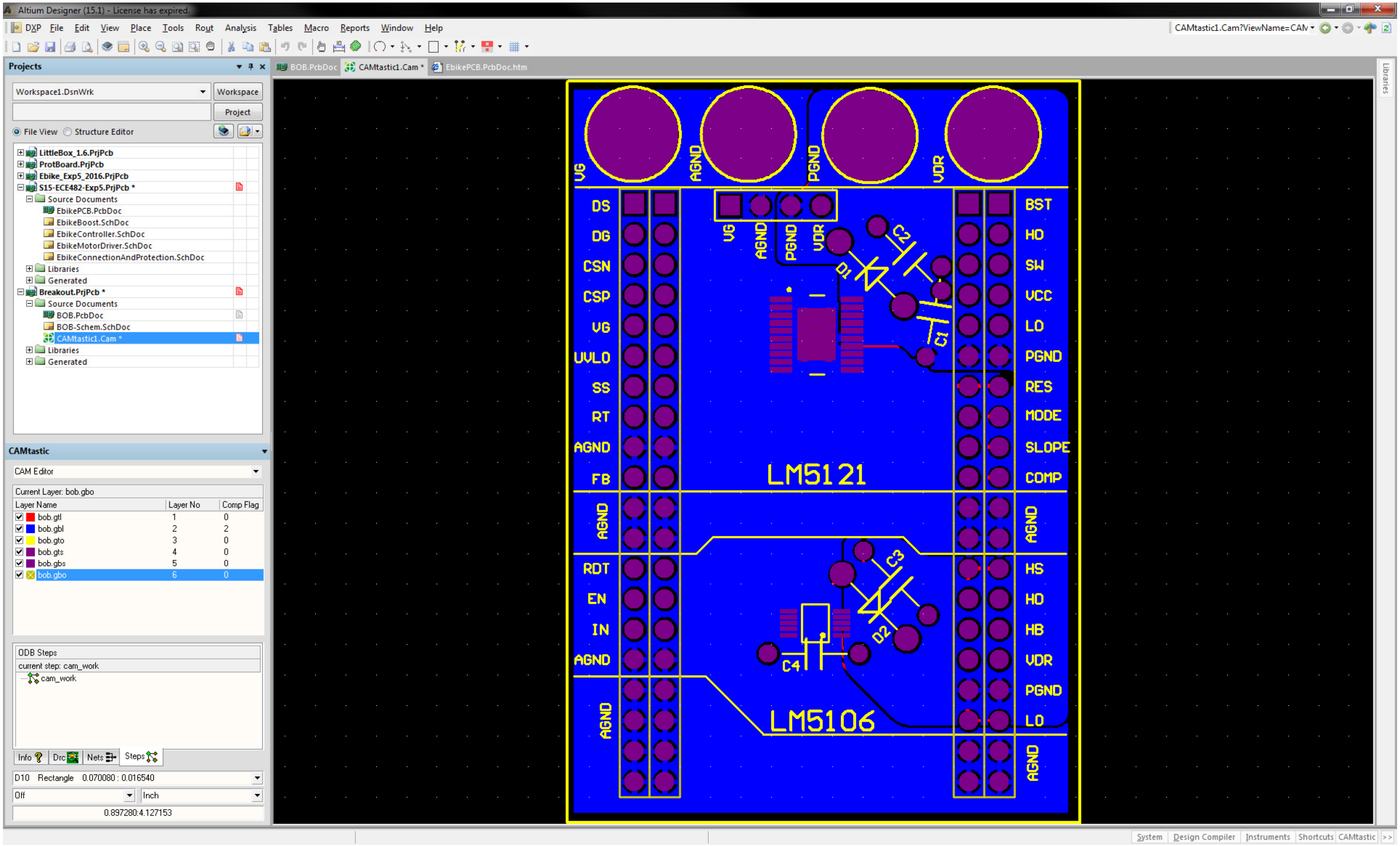
7. G1 – Signal Layer 1 – First (from top) inner layer copper
8. G2 – Signal Layer 2 – Second inner layer copper

   Only used in 4-layer board



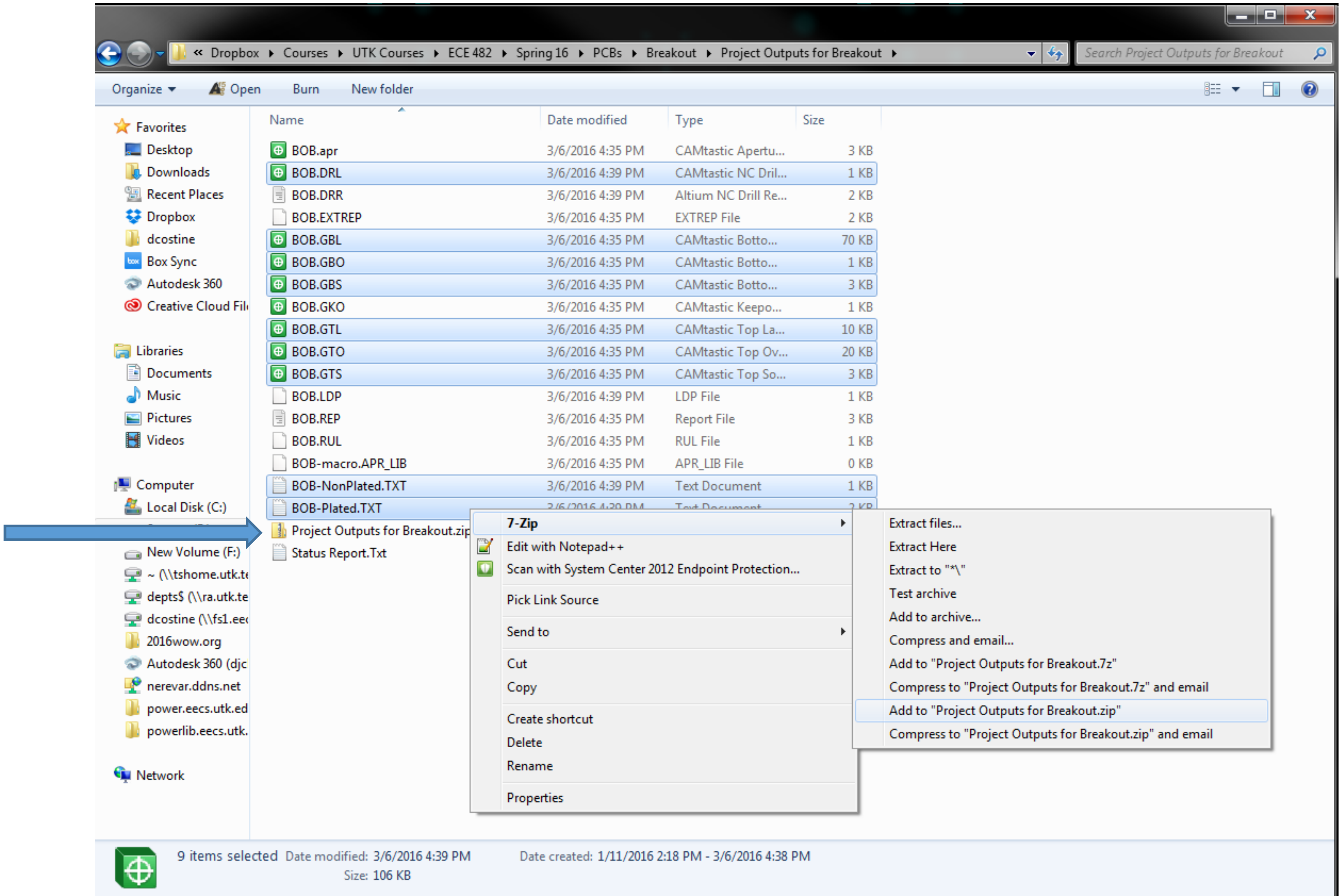
3. Use default values to generate the NC Drill File.

You may need to check the box to generate the actual .DRL file
Also, select the option to generate separate files for plated and non-plated holes.



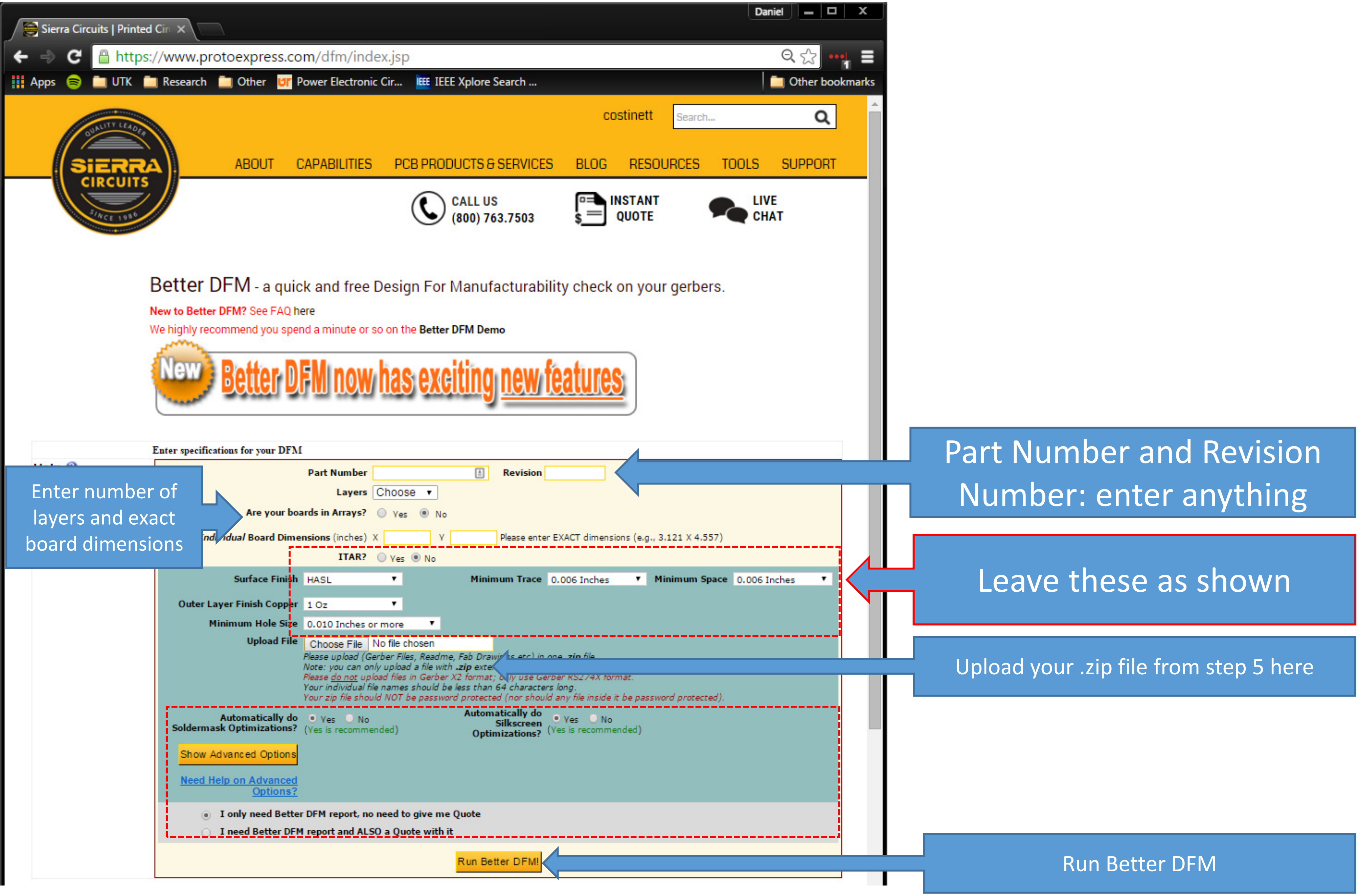
4. Review the camtastic files generated for both the gerber and NC drill process

Neither file is used, and may be closed without saving. However, they give a visual representation of what was generated and can be useful to see if any layers are misaligned, or text was not recognized.



4. Add all files to a .zip archive

- Include all of the gerber files from step 2, the DRL file and the two text files for plated and nonplated holes from step 3
- The files are located in your working directory (where everything in altium is saved) in a subfolder “Project Outputs for…”
- After you zip up all the files, immediately delete everything in the folder except the .zip archive.   This is done in case you need to resubmit later – it will prevent you from accidentally submitting a mix of updated and non-updated files.



5. In your web browser, navigate to https://www.protoexpress.com/dfm/index.jsp

You may need to register to create an account if you have not done so already. Later, you will need to share your username and password with our ordering staff. As such, it is recommended that you do not use a password you are uncomfortable sharing.

AutoMatch				
File Name	**File Type**	**Data Polarity**	**Layer Number**	**View**
bob-nonplated.txt	NC Drill	Positive		
bob-plated.txt	NC Drill	Positive		
bob.drl	Not Set	Positive		
bob.gbl	Bottom Copper Layer(Mixed)	Positive	2	
bob.gbo	Bottom Silkscreen	Positive		
bob.gbs	Bottom Soldermask	Positive		
bob.gtl	Top Copper Layer(Mixed)	Positive	1	
bob.gto	Top Silkscreen	Positive		
bob.gts	Top Soldermask	Positive		
< Back Sort Lyrs	Next >	Cancel/Delete		

6. On the next page, identify each layer

- The “automatch” button in the top-left can be used, but review the results.
- All layers should be positive polarity
- For four-layer boards, layer number should be 1-4 for the four copper layers, in order GTL, G1, G2, GBL
- It is easiest to select the non-plated and plated documents as your two NC Drill files, and leave the actual .DRL file off. It is still included in the files for the manufacturer’s reference.

Drill Information **Display Units:** Inch/Mils

Please help us make sure we have the correct drill sizes and plating types associated for the tools in your drill file(s). Setting of the finish size is only necessary if your drill file does not have the tool sizes defined within it. Click here to see some examples of drill files with the drill sizes defined so that they are detected automatically.

View File Contents
sanitized_ASDF/bob-nonplated.txt
sanitized_ASDF/bob-plated.txt
sanitized_ASDF/bob.drl
sanitized_ASDF/bob.gbl
sanitized_ASDF/bob.gbo
sanitized_ASDF/bob.gbs
sanitized_ASDF/bob.gtl
sanitized_ASDF/bob.gto
sanitized_ASDF/bob.gts
sanitized_ASDF.zip

Drill File: bob-plated.txt

Start Layer: bob.gtl

End Layer: bob.gbl

Tool Num	Finish Size(mil)	+ Tol.	- Tol.	Qty	Plating Type
1	17	3	3	11	PLT
2	33.46	3	3	8	PLT
3	35.43	3	3	84	PLT
4	50	3	3	4	PLT
5	210.63	3	3	4	PLT

Total Hit Count Qty: 111

Drill File: bob-nonplated.txt

Start Layer: bob.gtl

End Layer: bob.gbl

Tool Num	Finish Size(mil)	+ Tol.	- Tol.	Qty	Plating Type
6	30	3	3	1	NPT

Total Hit Count Qty: 1

< Back Submit Cancel

6. On the next page, identify each drill hit in your drill file(s) as plated or non-plated

- “Plating” means there will be metal connected from the top to bottom later through the hole. For conducting vias, they should be “PLT”. For holes which are not conducting, select “NPT”
- Submit when finished.

Better DFM Report

Print this page

What is Better DFM?

Daniel.Costinett
University of Tennessee

Better DFM # : afvbd-131798
Part Number : 482-S16

Report Date : 11th Jan 2016, 11:25 AM
Revision : 1

Feedback or Questions about this report or any particular issue in report? Feedback
Or you may send an email to BetterDFM@protoexpress.com

This report shows coordinates based on Datum (Origin) **as specified in the gerber files you uploaded.**
See location of Datum here.

Category of Issues	Number of Issues Found
Sierra Circuits recommends that customer should fix these issues	25 issues
Customer must review and fix (if needed) these issues	6 issues
Sierra Circuits will need customer approval to fix these issues	No such issues
Sierra Circuits will automatically fix these issues	22 issues

What if I can't or don't want to fix any of these issues that Sierra recommends?

Section 1 of 3:

Sierra Circuits recommends that customer should fix the issue(s) noted in this section

Layername: l1g

FileName: panelboard.gtl

X: 5.7993 **Y**: 4.8197

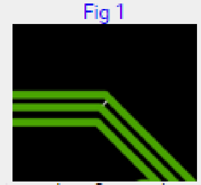
Value Found: 5.995 mils

Rule: 6.000 mils

Title:Circuit to circuit [Signal Checks]
(**Sierra Circuits recommends that customer should fix this issue**)

```
Circuits that are too close may bridge during imaging,
etching, plating, or soldering processes resulting in a
direct short. The circuit to circuit minimum spacing
being checked for is 6.000 mils. This location measures
5.995 mils.
```

Fig 1



This issue has **9** more locations.
Click image above to see locations.

7. After the filed are automatically checked, you will receive an e-mail with the results. To order the boards, you must have zero issues in “Sierra Circuits recommends that customer should fix these issues”

- For other categories, some of the issues may be false positives, or items that will be fixed automatically; it is rare to get a board with zero issues altogether.
- For any issues in the first category, you must alter the layout to remove them and resubmit.
- For all other issues, review the report and make sure you are confident that leaving them unfixed will not hinder board functionality.