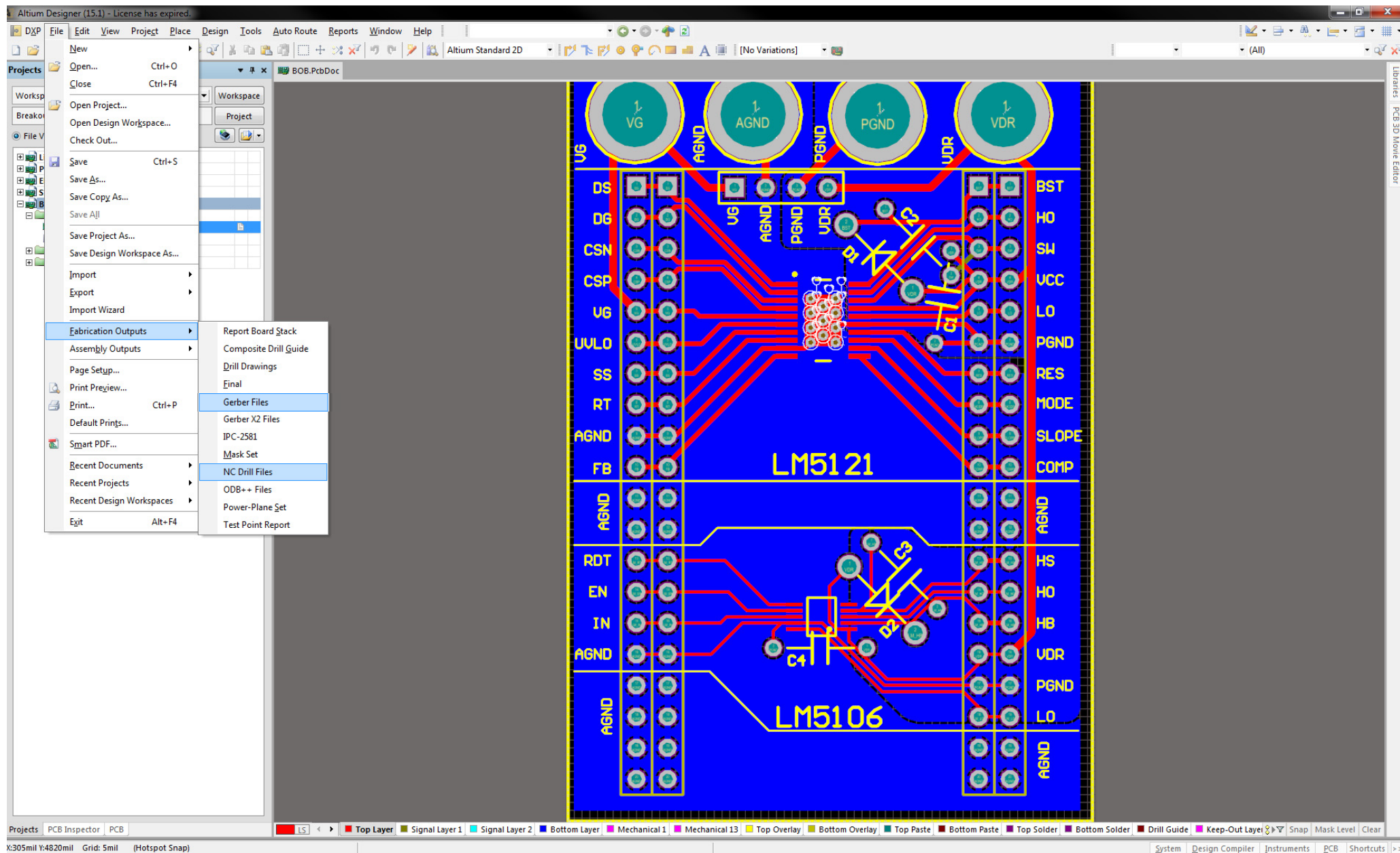
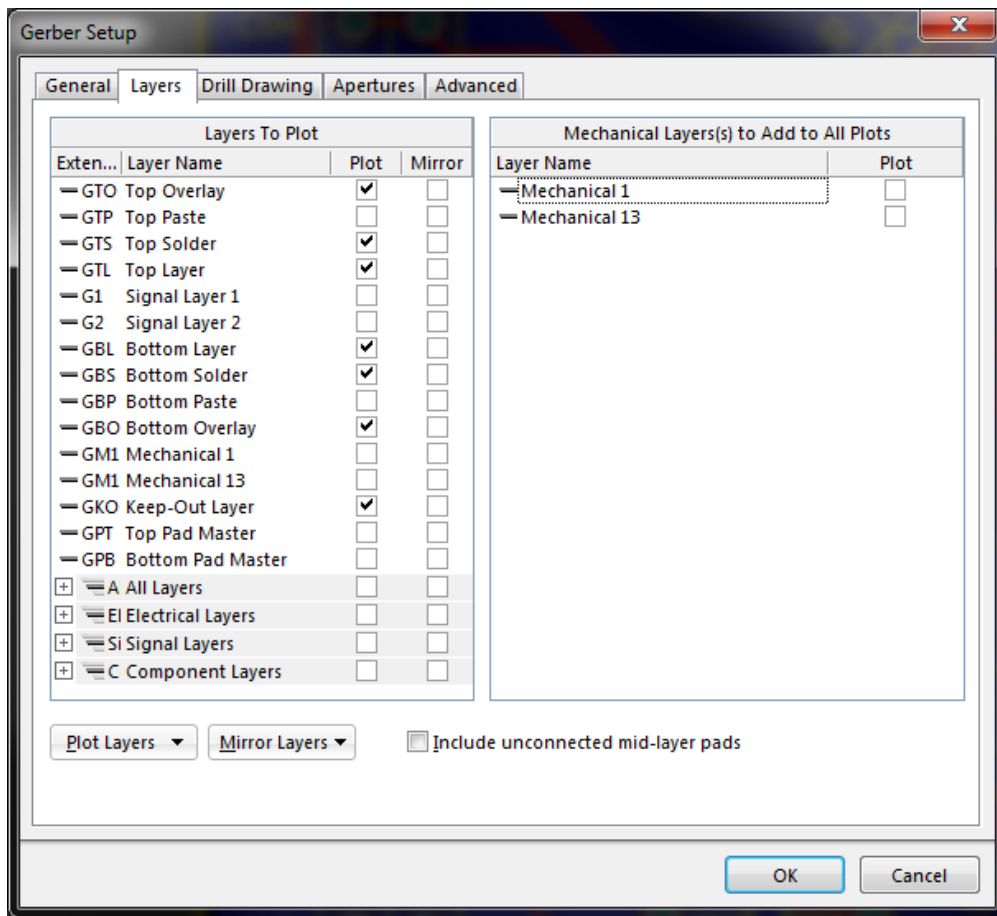


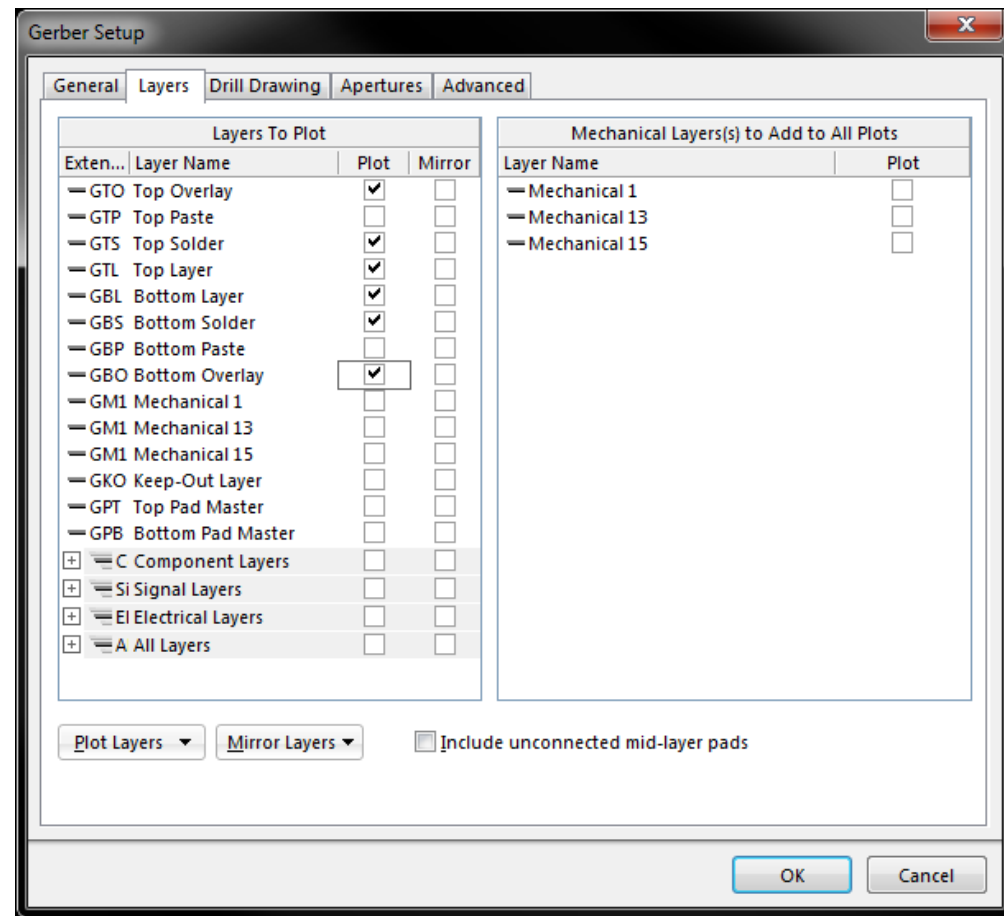
# Submitting PCBs in Altium to Sierra Circuits



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

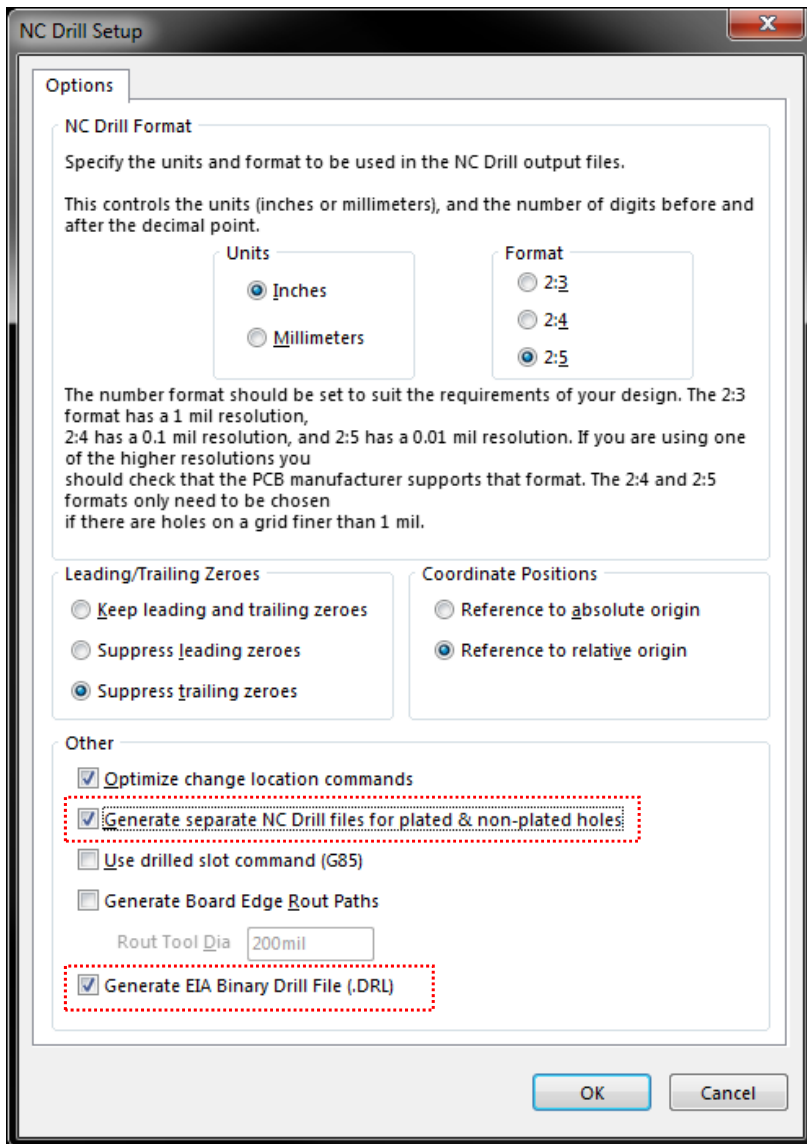


2-layer Gerber Template

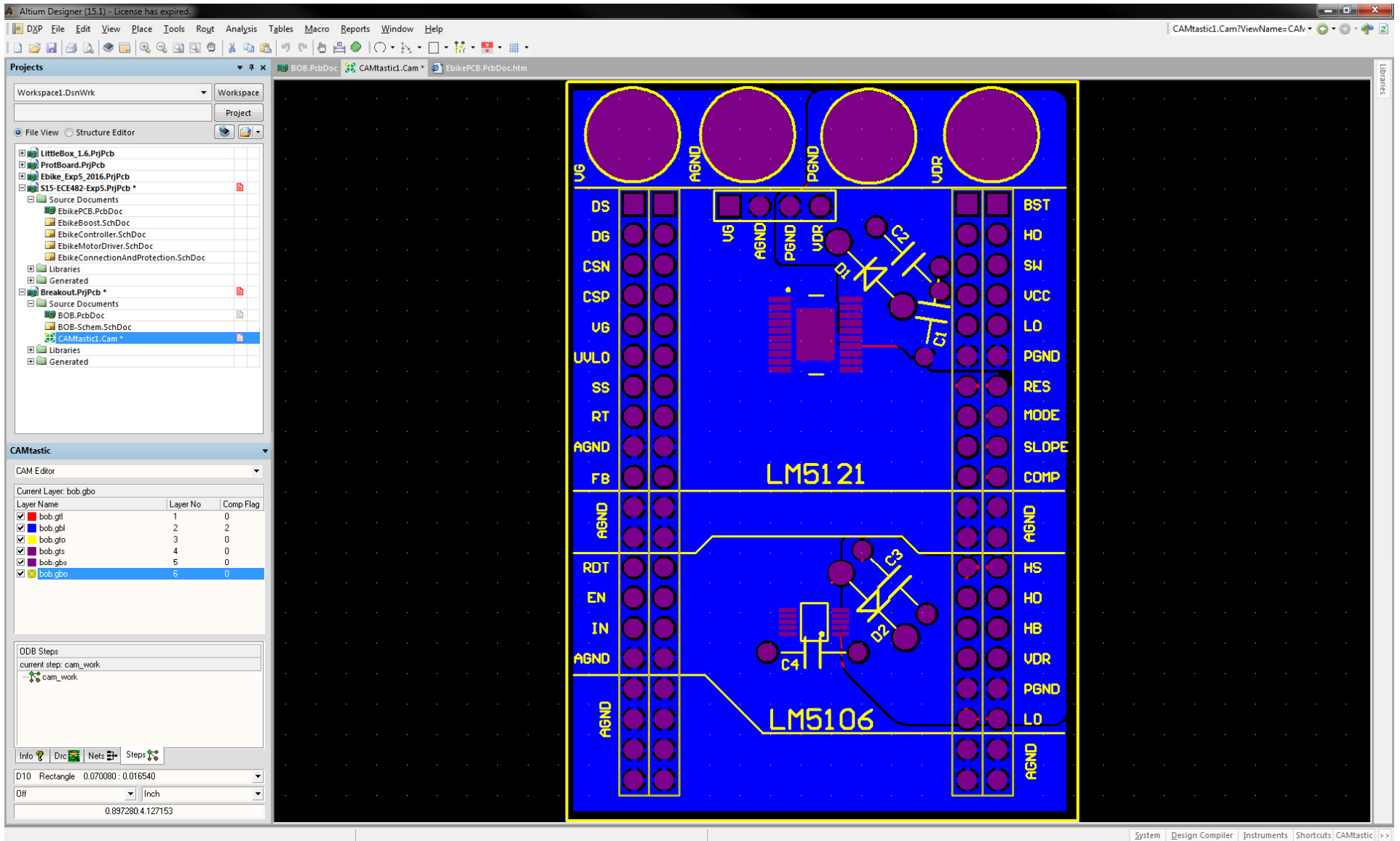
## 2. In the gerber file dialog, Select the layers needed for the PCB.

The necessary layers are:

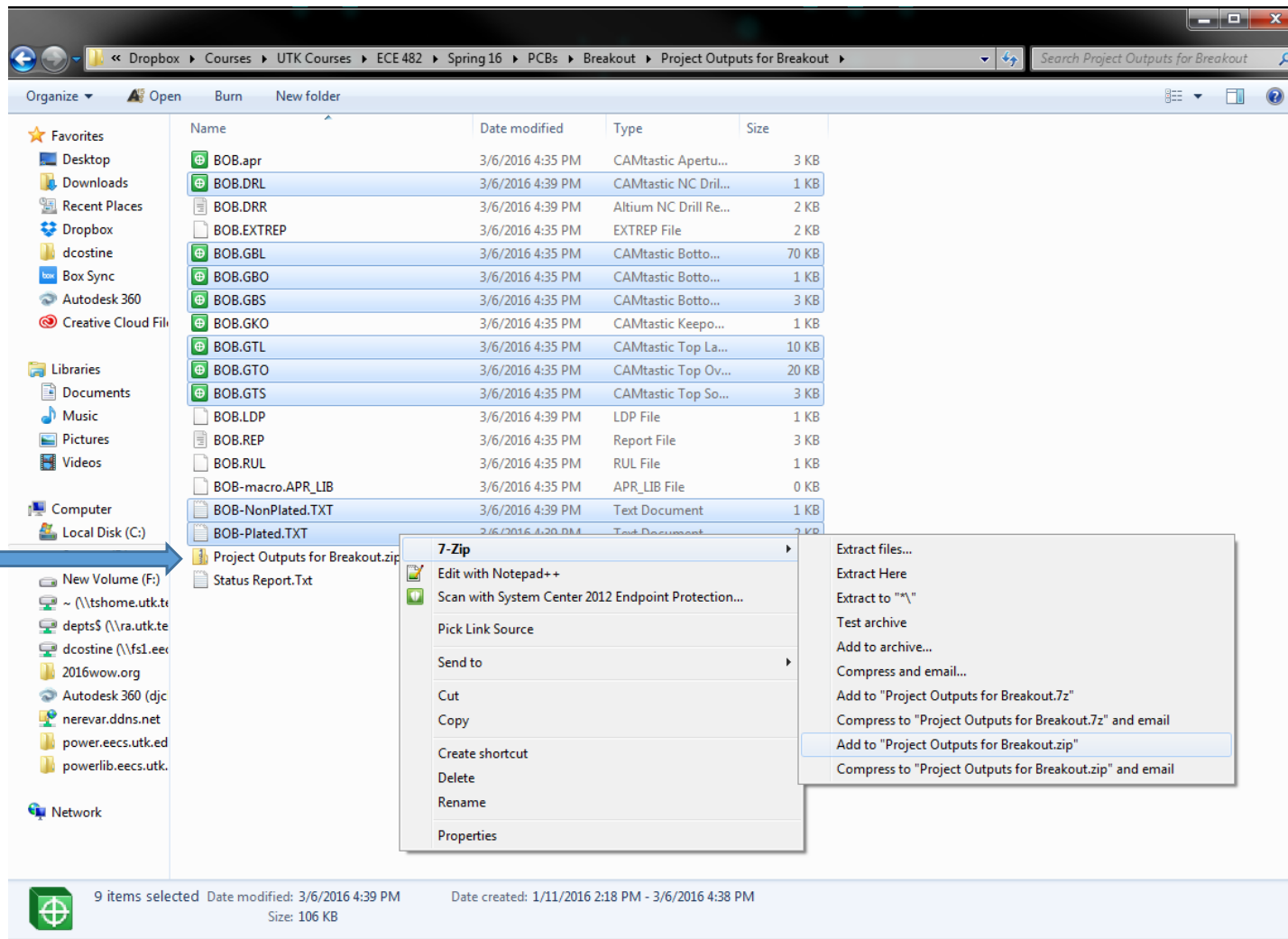
1. GTL - Top Layer – Copper on the top of the board
  2. GBL - Bottom Layer – Copper on the bottom of the board
  3. GTO – Top Overlay – Silkscreen Layer on the top of the board
  4. GBO – Bottom Overlay – Silkscreen Layer on the bottom of the board
  5. GTS – Top Solder– Soldermask Layer on the top of the board
  6. GBS – Bottom Solder– Soldermask Layer on the bottom of the board
  7. G1 – Signal Layer 1 – First (from top) inner layer copper
  8. G2 – Signal Layer 2 – Second inner layer copper
- } layer defines where soldermask *is not* present
- } 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.

costinett Search...

ABOUT CAPABILITIES PCB PRODUCTS & SERVICES BLOG RESOURCES TOOLS SUPPORT

CALL US (800) 763.7503 INSTANT QUOTE LIVE CHAT

Instant Quote: No Touch PCBs.  
 Now, RoHS-compliant (lead-free material and surface finish) [What is No Touch?](#)

Layers	6 layers	
True Turn Time (Business Days)	4 Days	Cut off time is 5 PM Pacific Time. When will my boards ship if I order today?
Quantity	2 2 boards	Select Quantity (up to 100 pieces).
Show additional quantities <a href="#">More</a>		
Board Dimensions	Please enter EXACT dimensions (e.g., 3.12 X 4.55)	
Minimum Finished Hole Size	15 mils (0.015"), Standard	
Minimum Trace / Space	6 mils (0.006"), Standard	No Touch now allows down to 4 mils trace/space!

Terms & Conditions And Product Specs [More](#) [Get Quote](#)

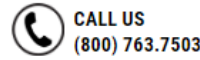
Enter exact board dimensions

Leave these as shown

Leave these as shown

Click Here

5. In your web browser, go to <https://www.protoexpress.com/orderProc/noTouchNew.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.



### Get Instant No Touch Quote (standard PCBs) [What is No Touch?](#) [Why buy from us.](#)

Layers	<b>6 Layers</b>	Min. Trace/Space	<b>0.006"</b>	Min. Hole Size	<b>0.015"</b>	Unit Price	<b>\$134.00</b>
Dimensions	<b>3" X 2"</b>	Turn Time	<b>4 Days</b>	Quantity	<b>2</b>	Total Price	<b>\$268.00</b>

[Proceed with this quote now >](#)

OR  
[Save this quote for future reference >](#)

Click Here

Choose No Touch when you need standard technology 2, 4 or 6 layer PCBs, **Lead-free FR-4 material**, **Lead-free HASL finish**, 0.062" thick, max 100 sq" area, 5 mils (0.005") or more trace/space, 8 mils or more (0.008") finished hole size, 1 Oz copper. 2-sided silkscreen and soldermask are included for FREE!

[See more Guidelines/Specs \(Download PDF\)](#)

If your board does not meet the NoTouch ordering specs, you can use our [Web PCB](#) online ordering or upload your files for a [Custom Quote](#).

You may select different Quantities & Turn Times from the table below. Click on price to select.

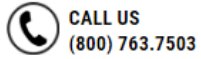
### Unit prices shown here

[\(Change to Lot Prices\)](#)

<b>True Turn Time</b> Quantity	<b>1 Days</b>	<b>2 Days</b>	<b>3 Days</b>	<b>4 Days</b>
2 Boards	\$1578.00	\$836.00	\$450.00	<b>\$134.00</b>
3 Boards	\$1067.00	\$619.00	\$330.00	\$96.00
5 Boards	\$658.00	\$378.00	\$218.00	\$86.00
10 Boards	\$342.00	\$191.00	\$131.00	\$82.00
15 Boards	\$218.00	\$126.00	\$95.00	\$69.00
25 Boards	\$126.00	\$72.00	\$61.00	\$53.00
50 Boards	\$62.00	\$35.00	\$33.00	\$29.00
100 Boards	\$35.00	\$22.00	\$18.00	\$16.00

[Need to see other quantity >](#)





Help For Layer Assignment Page  
Help For Drill Info Page

To ensure smooth processing, please do NOT press browser Refresh button or browser Back button on this page.



### Upload Files

We have now made No Touch service even better with some [new features!](#)  
Please provide us with part number and upload your design file.

Product Type **NoTouch**

Part Number \*  Rev.

Quantity 2 (Boards)

Quoted Area 6.00 Square inches

Exact Dimensions (X & Y) \* 3.0 by 2.0 inches

Upload Design file  No file chosen

Please upload your design in either Gerber RS-274X or ODB++ format.  
Send us files in **only one of these formats! Do not send both Gerber and ODB++ files.**

**For Gerber format:**  
Please load your gerber files in one .zip file.  
Note: in NO TOUCH PCBs service we do not read any fab drawings or readme files, so don't include those.  
Please do not upload files in Gerber X2 format; only use Gerber RS-274X format.  
Your individual file names should be less than 64 characters long.

**For ODB++ format:**  
Please load your file with .zip or .tgz extension.

Your zip file should NOT be password protected (nor should any file inside it be password protected).

**NOTE:**

**For Gerber File Submission**  
Next, you will go through 3 screens as part of file verification on your design files.  
1) Assign Layer information--which file is which layer  
2) Confirm Drill information  
3) Receive an Auto-File-Verification (AFV) number (like a confirmation number)

**For ODB++ submission**  
Next, you will go through 2 screens as part of file verification.  
1) Initialization screen since ODB++ already has file assignment info and drill Info all you do is press NEXT on this screen  
2) Receive an Auto-File-Verification (AFV) number (like a confirmation number)

To ensure smooth processing, please do not press Browser Refresh or Back buttons on the following 3 screens.



Please check settings in your

output files correctly.

Gerber files with following settings:

- Format = 274X
- Data Type = ASCII
- Coordinate = Absolute
- Zeros omitted = Trailing
- Number format = 2.4

Drill files with following settings:

- Format = Excellon
- Data Type = ASCII
- Units = Inch
- Tool Units = Inch
- Coordinate = Absolute
- Zeros omitted = Trailing
- Number format = 2.4

More Design Tips?

Questions? Need

During business hours call: 1-800-763-7503 X 9939 Or you can use our quick [customer service page](#)

Use whatever you like here

Upload your .zip file from step 5 here

Click Here

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

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
<a href="#">sanitized_ASDF/bob-nonplated.txt</a>
<a href="#">sanitized_ASDF/bob-plated.txt</a>
<a href="#">sanitized_ASDF/bob.dr1</a>
<a href="#">sanitized_ASDF/bob.gbl</a>
<a href="#">sanitized_ASDF/bob.gbo</a>
<a href="#">sanitized_ASDF/bob.gbs</a>
<a href="#">sanitized_ASDF/bob.gtl</a>
<a href="#">sanitized_ASDF/bob.gto</a>
<a href="#">sanitized_ASDF/bob.gts</a>
<a href="#">sanitized_ASDF.zip</a>

Drill File: bob-plated.txt						
Start Layer:	<input type="text" value="bob.gtl"/> ▼					
End Layer:	<input type="text" value="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:	<input type="text" value="bob.gtl"/> ▼					
End Layer:	<input type="text" value="bob.gbl"/> ▼					
Tool Num	Finish Size(mil)	+ Tol.	- Tol.	Qty	Plating Type	
6	30	3	3	1	NPT	▼
				Total Hit Count Qty: 1		

## 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



Daniel.Costinett  
University of Tennessee

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

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

[What is Better DFM?](#)

Feedback or Questions about this report or any particular issue in report? [Feedback](#)

Or you may send an email to [BetterDFM@protoexpress.com](mailto: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
<a href="#">Sierra Circuits recommends that customer should fix these issues</a>	25 issues
<a href="#">Customer must review and fix (if needed) these issues</a>	6 issues
<a href="#">Sierra Circuits will need customer approval to fix these issues</a>	No such issues
<a href="#">Sierra Circuits will automatically fix these issues</a>	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 files 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.