

What does FreeDFM Check For?

Complete ZIP Files 1. Gerber file for each copper layer 2. Gerber file for each soldermask layer 3. Gerber file for each silkscreen layer 4. Excellon or Gerber drill file	Inner Plane Layers 1. Spacing 2. Trace width 3. Annular ring 4. Inner clearance (drill to feature) 5. Thermal reduction
Outer Layers 1. Spacing 2. Trace width 3. Annular ring	Drill 1. Double hits 2. Missing hits
Inner Signal Layers 1. Spacing 2. Trace width 3. Annular ring 4. Inner clearance (drill to feature)	Soldermask 1. Undersized clearances 2. Missing clearances
Silkscreen 1. Line width	Upload Now at FreeDFM.com

FreeDFM Tips and Tools

1. If your drills do not line up correctly with your pads, then the automated system chose the incorrect scaling. Please send an email to FreeDFM.com with your quote number and request that they match the scaling on your drill file.

2. You may have multiple drill files, but FreeDFM currently only accepts one drill file. Please choose your plated through file to upload as the NC drill file. Make sure to load other drill files as other, because they will be needed later. You may also combine your drill files into one before uploading. CAM will merge all drill files when the order is placed. Drill files should be in Excellon (use the same scaling as other layers), EIA (ASCII), or Gerber format. Please note that with some layout software outputs the drill locations in one file and the drill sizes in another file, but in some cases FreeDFM can only accept one file, so drill report files can cause errors.

3. If your system generates drill files with the drill diameters embedded at the top of the drill file, then a drill tool report file is not required to be included within the zip file. Otherwise a drill tool report file is required. All NC Drill files should be exported in no less than 2.4 decimal coordinate placement. Drill files that have a smaller decimal format or a format that does not match the Gerber decimal format may suffer pad misalignment and then annular ring errors will be reported back to you. Some drill file examples below (as viewed in a text viewer):

Preferred format: Excellon with embedded drill sizes			
2.4 leading			
M48 T1C. 010 T2C. 125 % T1 X012345Y012345 T2 X022345Y022345 T00 M30	G90 M72 T1C0. 015748 X-395544Y-2039130 X-382650Y-2053382 X-379697Y-2053382 X-395544Y-2068027 X-309619Y-2053697 X-306666Y-2053697 X-293772Y-2039130 X-293772Y-2068027	% T1C. 015F095S3 X01185Y01255 X0142Y0132 X0142Y014 X03115Y0263 T2C. 03F197S55 X0165Y0153 X0166Y0118 X0172Y01835 T3C. 035F197S55 X0231Y0129 X0231Y0139	% T1C. 016 T2C. 024 T3C. 032 T1 X025801Y021671 X026201 X026601 X024651Y025221 X028251Y029071 Y029571 X029701Y028471 X030101

Occasionally Acceptable formats:

Gerber RS-274X (may not have plating info)	ASCII formats (i.e. Camtastic)	Excellon format with commented drill table (sizes and plating)
* %FSLAX24Y24*% %MOIN*% %ADD10C,0.010000*% %ADD11C,0.015000*% %ADD12C,0.019690*% %ADD15C,0.037000*% %ADD19C,0.116000*% %IPPOS*% %LNdrp.gbr*% %LPD*% G75* G54D10* X016350Y022450D03* X018350Y035850D03* X017500Y036150D03* X017950Y039500D03* X014150Y039600D03* X019000Y039500D03*	T_ €T_ €#_v 2 s k_ €#_v 2 s k_ €#_v 2 s k_ €#_v 2 s k_ €#_v 2 s k T_ €T_ €#_v 2 s k_ €#_v 2 s k_ €#_v 2 s_k_ €#_v 2 s_k_ _€#_v 2 s_k_ €#_v 2 s_k_ €#_v 2	;LEADER: 12 ;HEADER: 563-81498-001 Rev A ;CODE : ASCII ;FILE : ncdrill1 for layers TOP and BOTTOM ;Holesize 1. = 13.000000 PLATED MILS ;Holesize 2. = 38.000000 PLATED MILS ;Holesize 3. = 42.000000 PLATED MILS G90X02200Y09960X02360Y09960X01630Y10500X01630Y09450 M00X-00250Y09950X-00650Y09950 X00150Y09800

Occasionally Acceptable: Excellon format with a separate drill file (table)

Excellon format, but without embedded drill sizes	A separate file with the drill (tool) sizes (in this case, drill.rep)
% T1F095S300 X038552Y030071 X046502Y025733 X049503Y025856	Drill Sizes Report ===== Tool Size Pltd Feed Speed Qty ==== ===== 1 10 x 95 300 345 2 12 x 95 300 2

X052041Y030292	3	35	x	197	550	211
X055527Y025968	4	37	x	197	550	80
X058046Y025292	5	37.01	x	197	550	14
X061126Y025277	6	68.9	x	89	550	20
X065095Y025135	7	75	x	81	550	9
X072170Y025404	8	91	x	66	550	2
	9	110	x	69	658	2
	10	125	x	35	794	5

4. Multiple via drill diameters that have their pads covered in soldermask will report as errors because the software does not know if you want all your vias covered (tented) or not. You can either clear soldermask from (give relief to) the via pads, or ignore this error.

5. Fabrication drawings with detailed information for Slots, cutouts, and outlines will not appear in your output when they are uploaded as other. If you desire to see the shape or outer contour of your PCB and any internal cutouts, you can include these outlines on your solder mask layers for visual reference during your FreeDFM image plot review. All files from FreeDFM are viewed from the topside of the board, through the board.

The bottom side silkscreen text should look backwards for it to read correctly when viewed from the bottom.

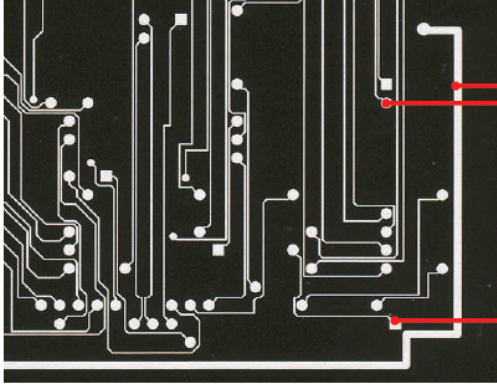
6. If your board outline is on a separate layer and uploaded to other, it will not be shown on the FreeDFM output, nor will tests be run using your outline. Please note: If exposed copper is not desired on the board edge, then copper should be kept 10 mils away from outline rout edge by outer layers and 15 mils for inner layers. For scoring keep copper 15 mils away for outer layers and 20 mils for inner layers. Please also note: If an outline is not included in one of your uploaded layers, and an order is placed without one, the job will go on hold until one is provided. The GKO (keep out) layers can only be used as the board outline if it is clearly indicated that we can do so.

7. FreeDFM tests are designed for standard and custom specification orders, and it will not flag errors that may occur if the job is ordered as any other type. For instance the copper to copper spacings (trace and space) for other order types differ from what is allowed on standard and custom spec orders. A 33 special order is allowed a minimum trace and space of 6 mils for 1 oz copper, but FreeDFM will not flag trace and spacings between 5 and 6 mils.

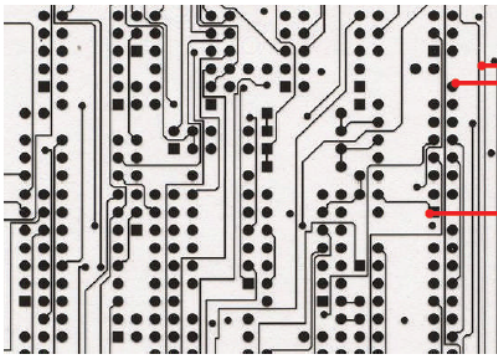
8. When selecting the image polarity of a layer, then like a photograph, choose positive if your inner layer is viewed literal and negative if it is the opposite of where copper should be.

9. When assigning inner layer polarity on the FreeDFM system, one needs to know if the inner layers are drawn with traces or filled with copper pour and or even both. This type of layer is a positive image. It will look just like an outer layer. The reverse and or negative polarity is when the inner layer is a power or ground flashed plane with anti pads for drill clearance and flashed thermal spoke wheel pads for inner layer drill connection.

Incorrectly assigning the inner layer polarity will result in erroneous error reports.



Inner layer film is “negative”. That means that the copper patterns left behind after processing the core are the “clear” areas on the film.



Outer layer film is “positive”. The traces and pads that are “opaque” on the film are copper on the outside of the board and the clear areas will be clear of copper,