

PCB Design e-file (Example)

Software information

PADS:

Encourage users to use PADS software provided by TSRI, with template, consulting, e-Learning, and winter / summer vacation training courses.

Altium:

If the user use Altium software, please use versions after [2012 \(including\)](#). Do not use the version before 2012. Because the former version might not compatible with TSRI's template, and generates too many DRC errors. If the user wants to use the version before 2012, please set up the Design Rules by oneself and make sure to comply the DRC listed in TSRI's web Design Rule manual.

Otherwise, the application might be rejected because of DRC errors. After

[After 2015/03, there is no update for the Altium template, please use PADS software instead.](#)

Other Software:

No template. If the user wants to use the version before 2012, please set up the Design Rules by oneself and make sure to comply the DRC listed in TSRI's web Design Rule manual.

[1] Name and Phone no. of Applicant

PCB Recognition Code [#]	2L_112A_1A2B
Applicant Name	Jack Smith
Applicant Phone No.	0912345678 、 03-5773693 # 999
PCB Layout software name /version	PADS VX.2.5

PCB Recognition Code Naming rule:

Process Code_year batch_self naming characters (4 digits of English Characters or numbers).

ex: 2L_112A_1A2B, marked at "Silkscreen top" upper left corner with a frame to enclose the code.

[2] Topics

Topics in English	A 25GHz Current Reused Differential Oscillator PCB Board Design
-------------------	---

[3]PCB Gerber Layer Naming

Reference: ["TSRI CB Fabricaiton Design Rule Manual"](#)

Item	Remarks
Each Layer	Generate Gerber file separately, Upload separately, and record the file name in "user naming" column of the following table.
*mark	Necessary files for upload. All the Gerber files and DRC report are necessary files for upload.
"Other" Column	If there is no other necessary file, skip this column. Altium slot layer upload here. If there are more than 1 file, please compress them. Use .zip or .7z format.

FR4 2-Layer Board / RO4003 2-Layer Board

Fill in the following table:

No.	Layer Name in Chinese	Layer Name in English	Recommend Naming [#]	User Naming 1 (PADS)	User Naming 2 (Altium)
1*	外部金屬層_Top	Top	TOP.---	my_design_TOP.pho	my_design.GTL
2*	外部金屬層_Bottom	Bottom	BOT.---	my_design_BOT.pho	my_design.GBL
3*	防焊層_Top	Solder Mask Top	SMT.---	my_design_SMT.pho	my_design.GTS
4*	防焊層_Bottom	Solder Mask Bottom	SMB.---	my_design_SMB.pho	my_design.GBS
5*	文字層_Top	Silkscreen Top	SST.---	my_design_SST.pho	my_design.GTO
6*	文字層_Bottom	Silkscreen Bottom	SSB.---	my_design_SSB.pho	my_design.GBO
7*	鑽孔層	NC Drill	NCD.---	my_design_NCD.drl	my_design.TXT
8*	板框層	Board Outline	BO.---	my_design_BO.pho	my_design.GKO
9*	機械加工層	Mechanical	ME.---	my_design_ME.pho	my_design.GM1
10*	鑽孔孔徑資料檔	NC Drill Table	DW.---	my_design_DW.pho	my_design.GD1
11*	DRC 報告	DRC		my_design_clear.lst	my_design.html
12	其他	Other		my_design_DFF.lst	my_design_slot.TXT (if any)

#: File extension depends on software

FR4 4-Layer Board / RO4003 4-Layer Board

Fill in the following table:

No.	Layer Name in Chinese	Layer Name in English	Recommend Naming [#]	User Naming 1 (PADS)	User Naming 2 (Altium)
1*	外部金屬層_Top	Top	TOP.---	my_design_TOP.pho	my_design.GTL
2*	內部金屬層 L1	Inner_1	IN1.---	my_design_IN1.pho	my_design.G1
3*	內部金屬層 L2	Inner_2	IN2.---	my_design_IN2.pho	my_design.G2
4*	外部金屬層_Bottom	Bottom	BOT.---	my_design_BOT.pho	my_design.GBL
5*	防焊層_Top	Solder Mask Top	SMT.---	my_design_SMT.pho	my_design.GTS
6*	防焊層_Bottom	Solder Mask Bottom	SMB.---	my_design_SMB.pho	my_design.GBS
7*	文字層_Top	Silkscreen Top	SST.---	my_design_SST.pho	my_design.GTO
8*	文字層_Bottom	Silkscreen Bottom	SSB.---	my_design_SSB.pho	my_design.GBO
9*	鑽孔層	NC Drill	NCD.---	my_design_NCD.drl	my_design.TXT
10*	板框層	Board Outline	BO.---	my_design_BO.pho	my_design.GKO
11*	機械加工層	Mechanical	ME.---	my_design_ME.pho	my_design.GM1
12*	鑽孔孔徑資料檔	NC Drill Table	DW.---	my_design_DW.pho	my_design.GD1
13*	DRC 報告	DRC		my_design_clear.lst	my_design.html
14	其他	Other		my_design_DFF.lst	my_design_slot.TXT (if any)

[#]: File extension depends on software

[4] Gerber Output Setup / Figure

Please make sure the uploaded Gerber and Drill files are correct, and the same as those shown in the attached figures.

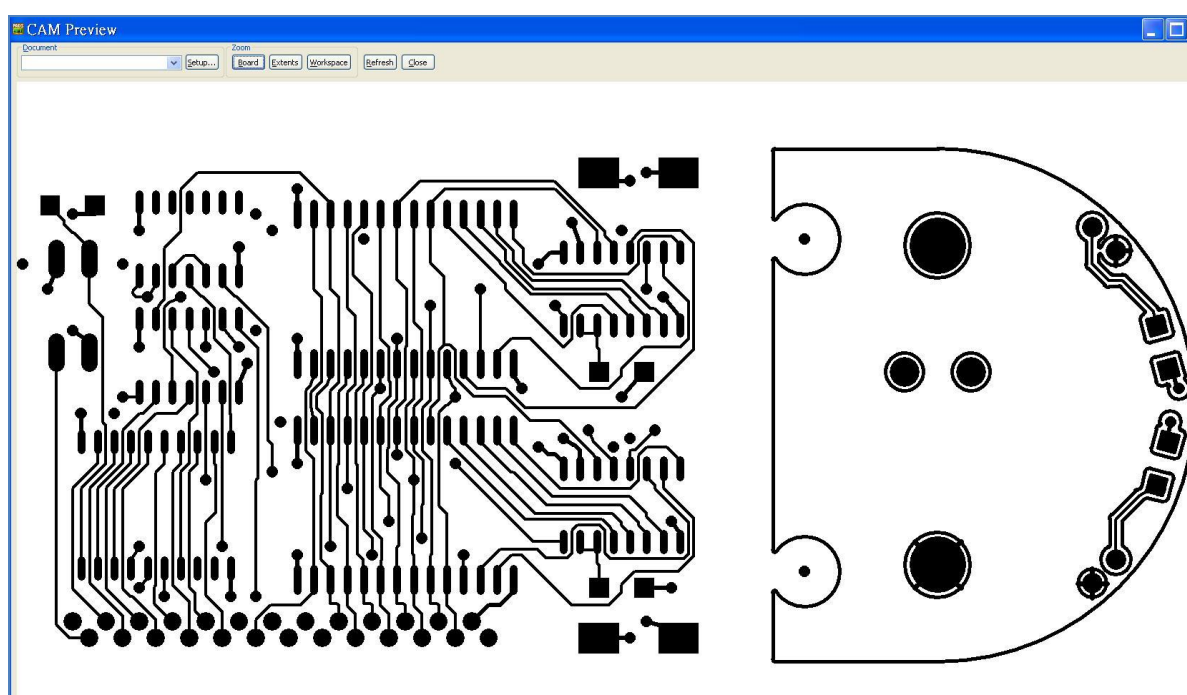
Altium: Open .GTL 、.TXT and .GD1 files, and attach the figures separately.

PADS: Use command **File > CAM >** select correct **Document Name > Preview** (click **Board or Extents**), and attach the figures separately.

Other Software: Use similar method.

PCB Size (Length x Width , mm x mm)	<u>100</u> mm x <u>50</u> mm
Output Units	<input type="checkbox"/> Inches (English)
	<input checked="" type="checkbox"/> Millimeters (Metric)
Format (I:D)	Integral (Leading) =3
	Decimal (Trailing) =3
Coordinates	<input checked="" type="checkbox"/> Absolute
	<input type="checkbox"/> Incremental
	<input type="checkbox"/> Not applicable
Zeros suppression	<input checked="" type="checkbox"/> Leading Zeros suppression
	<input type="checkbox"/> Trailing Zeros suppression
	<input type="checkbox"/> None (Keeping leading and trailing zeros)

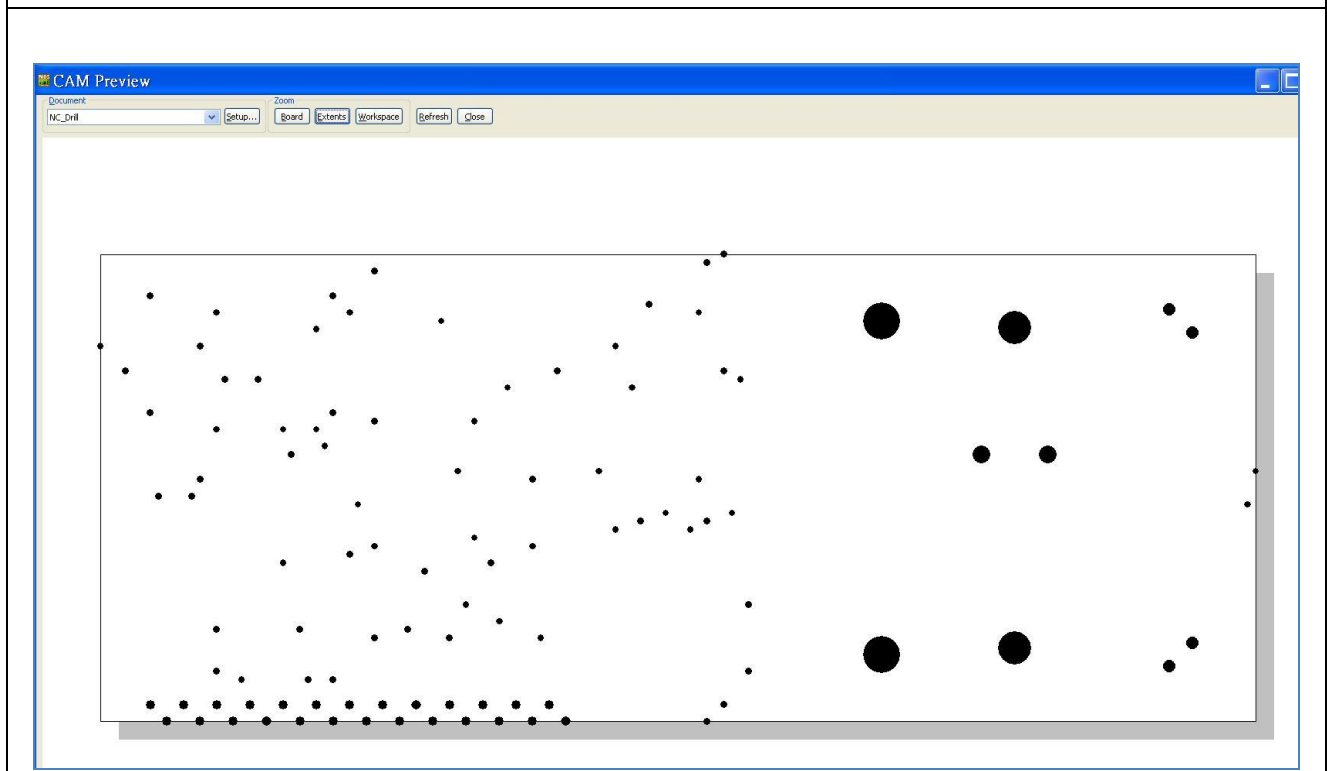
Gerber figure (Top) (Attach below.)



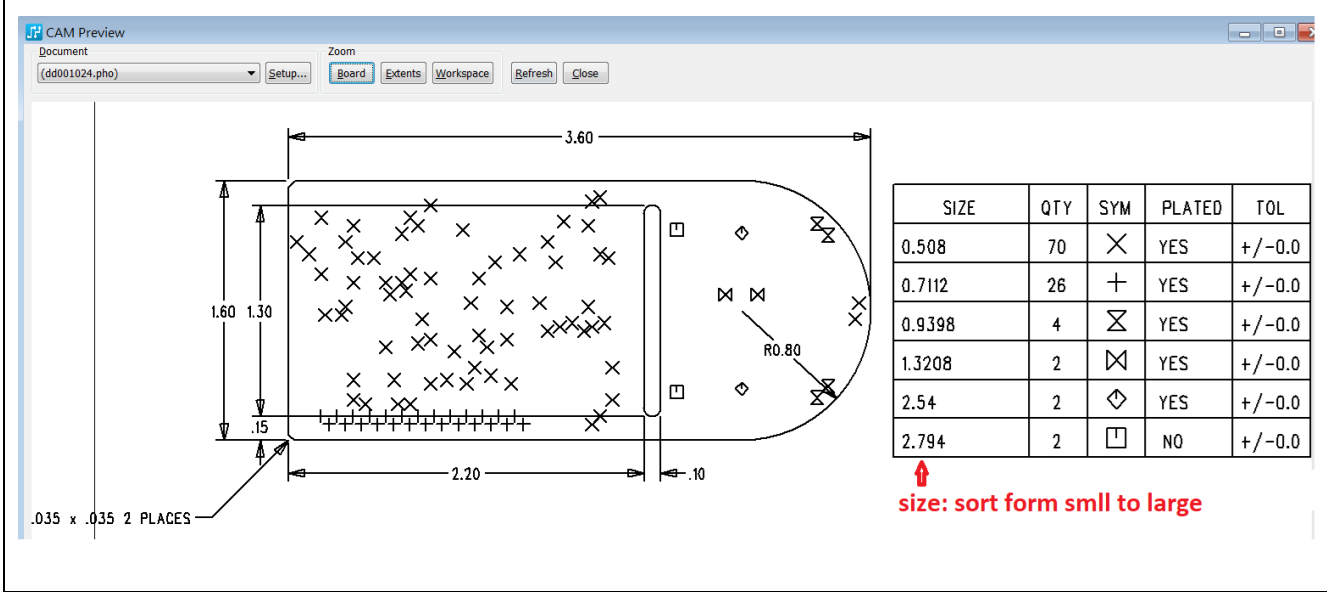
[5] Drill Output Setup / Figure

Output Units	<input type="checkbox"/> Inches (English)
	<input checked="" type="checkbox"/> Millimeters (Metric)
Format (I:D)	Integral (Leading) = 3
	Decimal (Trailing) = 3
Coordinates	<input checked="" type="checkbox"/> Absolute
	<input type="checkbox"/> Incremental
	<input type="checkbox"/> Not applicable
Zeros suppression	<input checked="" type="checkbox"/> Leading Zeros suppression
	<input type="checkbox"/> Trailing Zeros suppression
	<input type="checkbox"/> None (Keeping leading and trailing zeros)

Drill Figure (Attach below.)

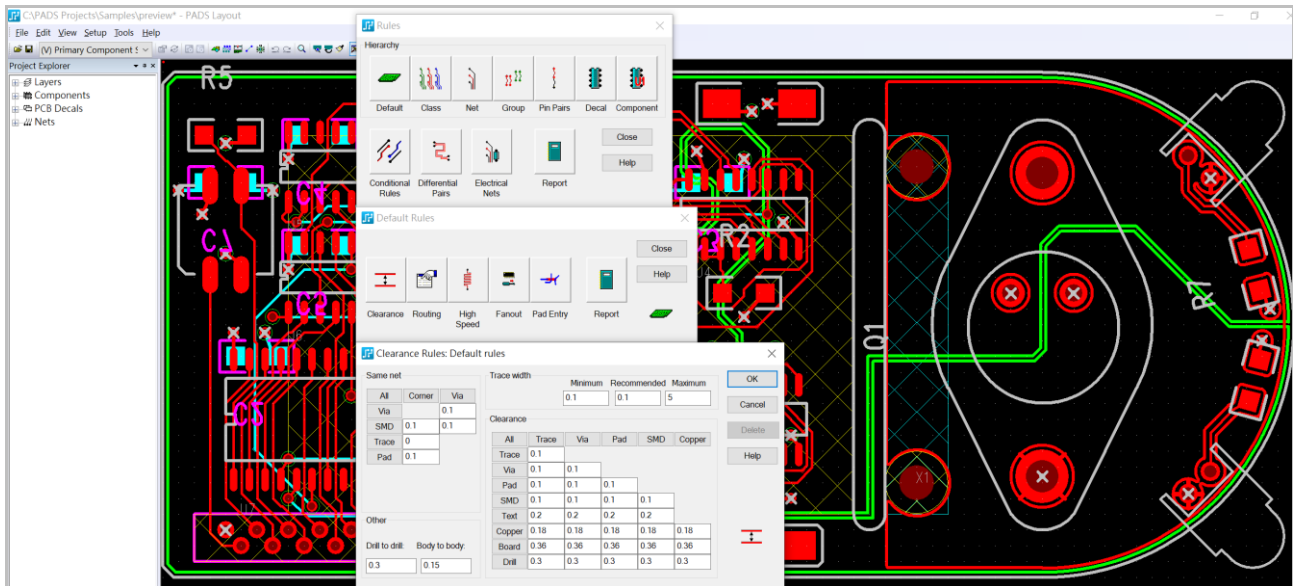


Drill Table (Attach below.)

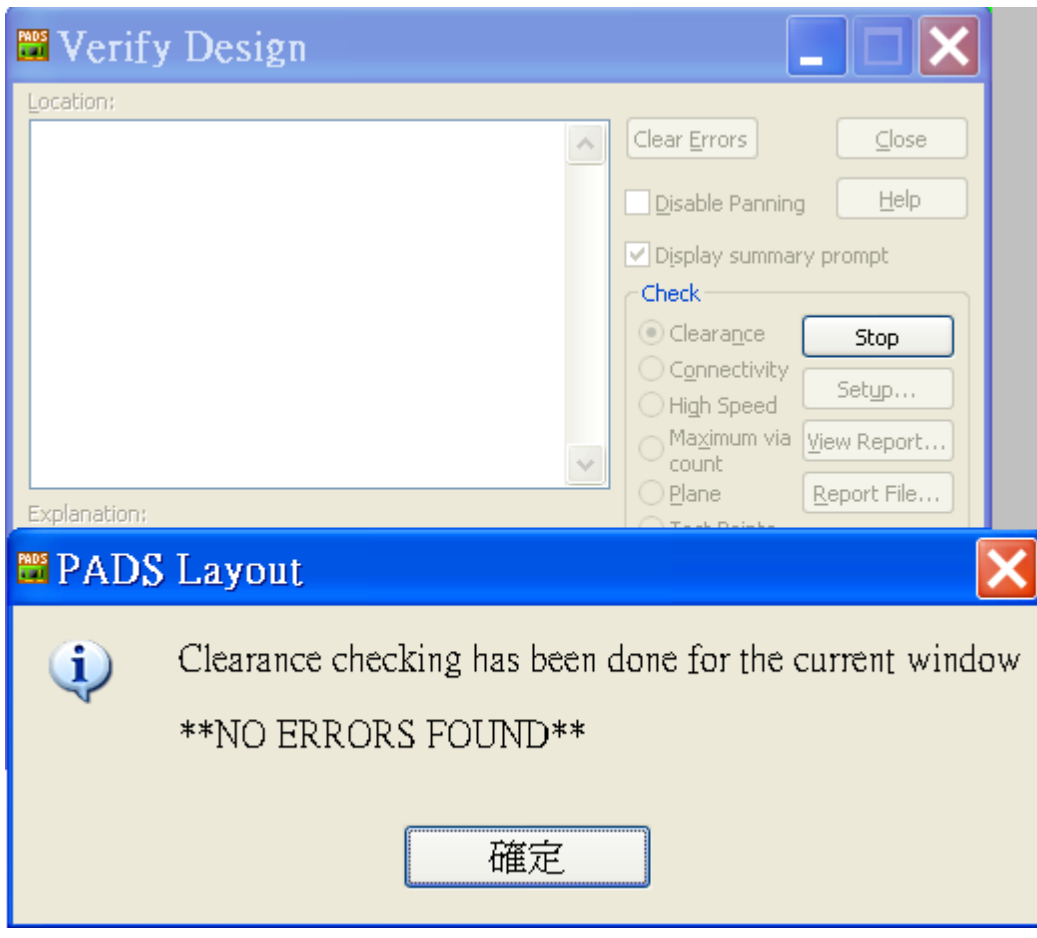


[6] PCB DRC Verification Results Figures

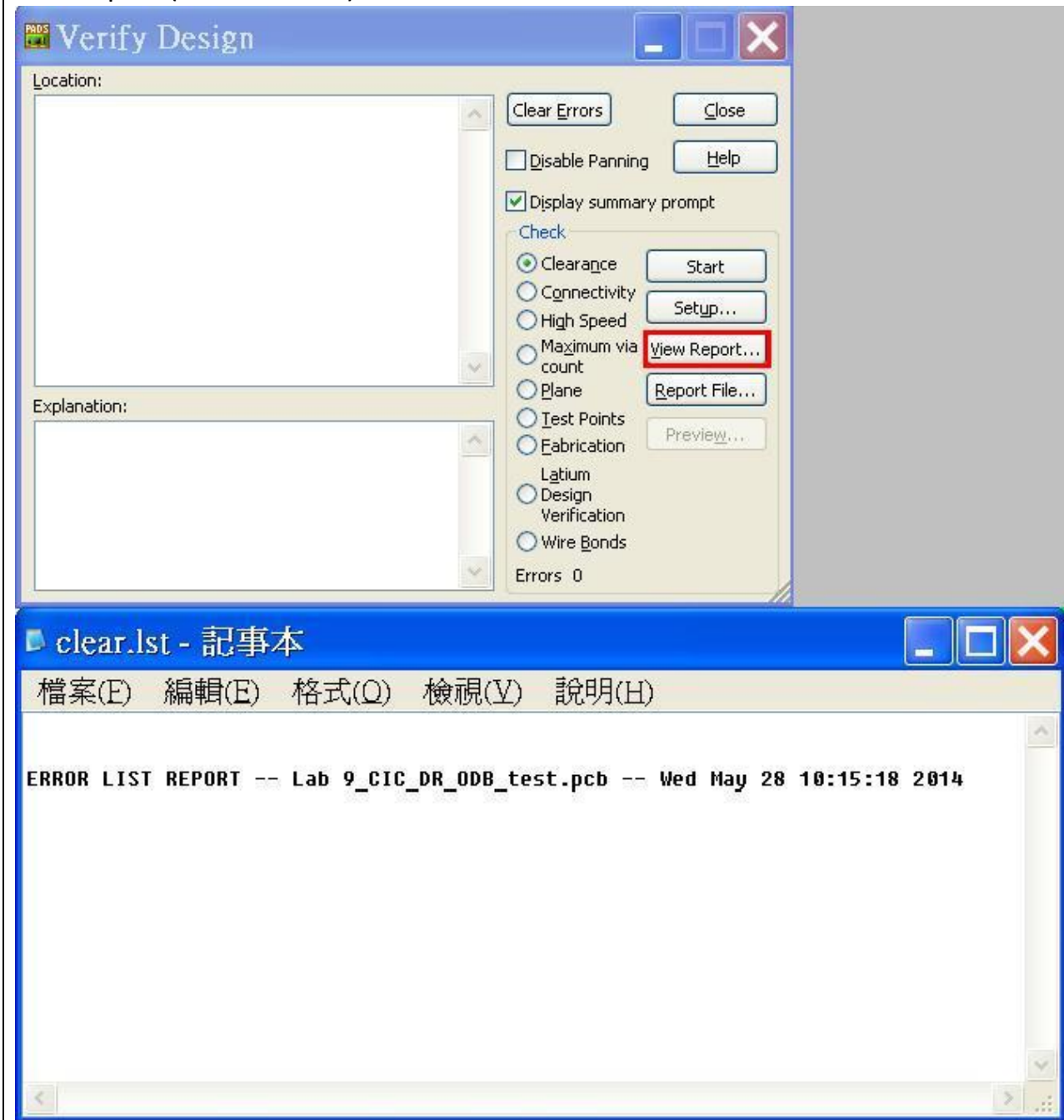
No DRC error figure from software.



After DRC Verification, no error figure. (ex: Clearance)



DRC Report (ex: Clearance)



[7] Assembly (2D or 3D)

Display: (PADS) Use command **View > PADS 3D** to display the layout, and attach figure below.

Part height setup: (PADS) **Project Explorer > PCB Decals > (Select one Decal) > RMB (Right Mouse Button) > Properties Attribute > Add > Geometry. Height**, or when building part Library, the setup of **Attributes**.

After these setup, user can use command **Edit > Attribute Manager > Components > Attribute > Show > Geometry. Height** to verify.

