

Cadence/Orcad to Adiva Interface (Quick-Start User Guide)

Notice

Representations in this User Guide are meant as an overview and quick reference. Full details can be found in the On-Line manuals located at the *ADIVA Corporation* website - www.adiva.com

All rights reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted, in any form or by any means - electronic, mechanical, photocopying, recording, or otherwise - without the prior written permission of *ADIVA Corporation*.

ADIVA Corporation provides this User Guide "as is", without warranty of any kind, either expressed or implied, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. *ADIVA Corporation* may make improvements and/or changes in the product (s) and/or the program (s) described in this manual at any time and without notice.

Although *ADIVA Corporation* has gone to great effort to verify the integrity of the information herein, this publication could contain technical inaccuracies or typographical errors. Changes are periodically made to the information herein. These changes will be incorporated in new editions of this publication.

Preparing Cadence/Orcad

- Make sure **Layer Names** in the Cadence/Orcad stackup **DO NOT** contain spaces.
- Cadence/Orcad's Artwork Control form needs to be filled in and ready for Gerber output. **Do not** manually create Gerber/Drill in Allegro – the ADIVA Interface will instruct Cadence/Orcad to do that automatically.
- Under **Artwork Control Form > General Parameters** in Allegro - make sure that units, format and polarity are set appropriately. **Undefined line width must be set to a size greater than “0”**. Acceptable data formats are Gerber 274X or Gerber 6x00 (standard 274D).
- It is helpful to have an “Outline” layer containing ONLY the board outline as a defined artwork. Do not include a title block on the board outline artwork otherwise title blocks will be included in checking. **Make sure the value for Undefined Line Width is set to a size greater than “0”**.
- If using the **ads_sdart** variable, the Skill program will look there for parameter files controlling data output. It will also place all created files (ie: Gerber, drill, etc) into the **ads_sdart** location – as well as – a “DRC_jobname” directory created under the Cadence/Orcad .brd file.

The **ads_sdart** variable if set in Cadence/Orcad, must also be set as a Windows environment variable pointing to the same directory.

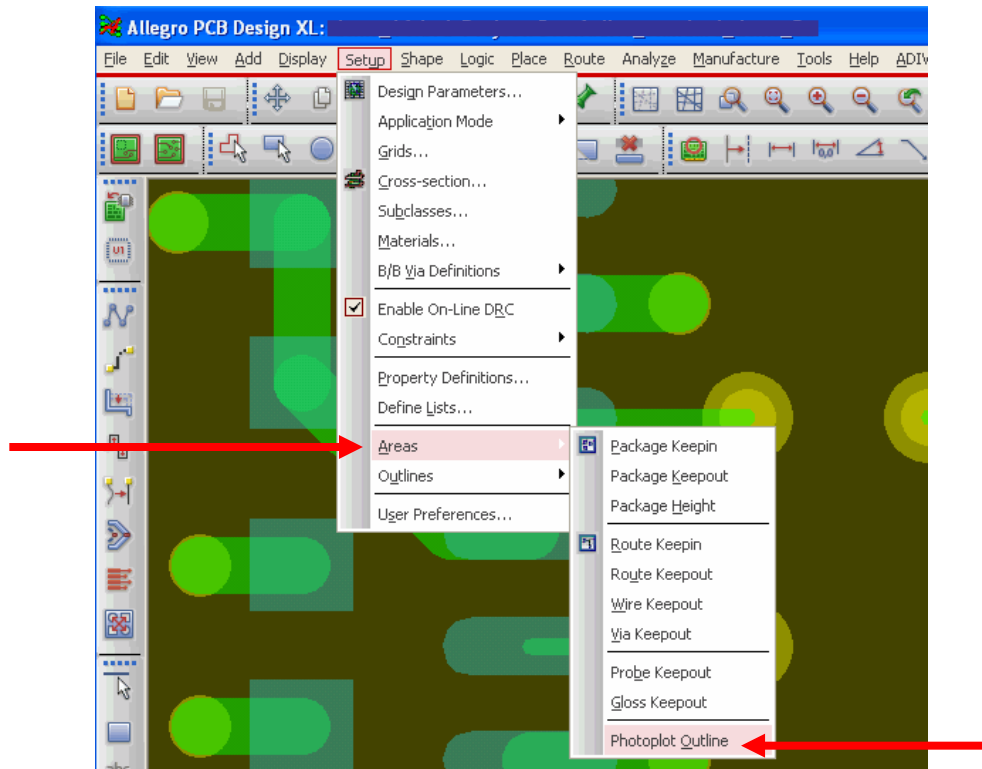
- If using the **art_path** and / or **ncdpath** variables, the Skill program will follow the paths set by these variables for the parameter files **art_param.txt** and **nc_param.txt**. First file found is used. It will also place all created files (ie: Gerber, drill, etc) into the .brd file directory – as well as – a “DRC_jobname” directory created under the Cadence/Orcad .brd file.
- If no variables are used, the Skill program will look locally (.brd file location) for parameter files. It will also place all created files (ie: Gerber, drill, etc) into the .brd file directory – as well as – a “DRC_jobname” directory created under the Cadence/Orcad .brd file.

Preparing Cadence/Orcad

Very Important:

If negative plane layers are used in the design and Gerber 274X output is intended, make sure there is a **Photoplot Outline** defined that is slightly larger than the board outline – this prevents Allegro from creating large polygons filling the entire film size outside the board outline on negative plane layers.

*** Inclusion of **Photoplot Outline** will have a direct – **positive** - impact on Adiva software performance when used.

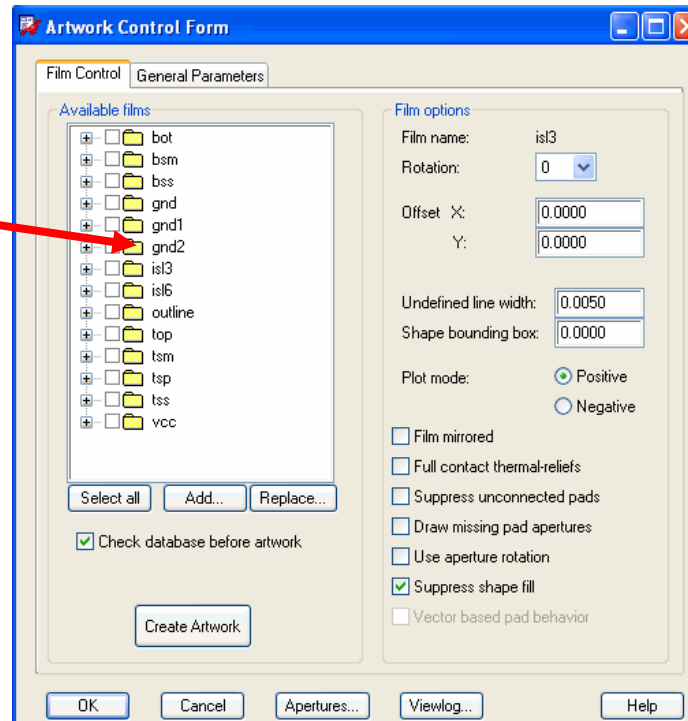


Preparing Cadence/Orcad

Very Important:

Artwork film-names must **NOT** contain file extensions. Cadence/Orcad will generate artwork automatically with the file extension “.art”.

Note simple film-name format
Do not assign film-name extensions



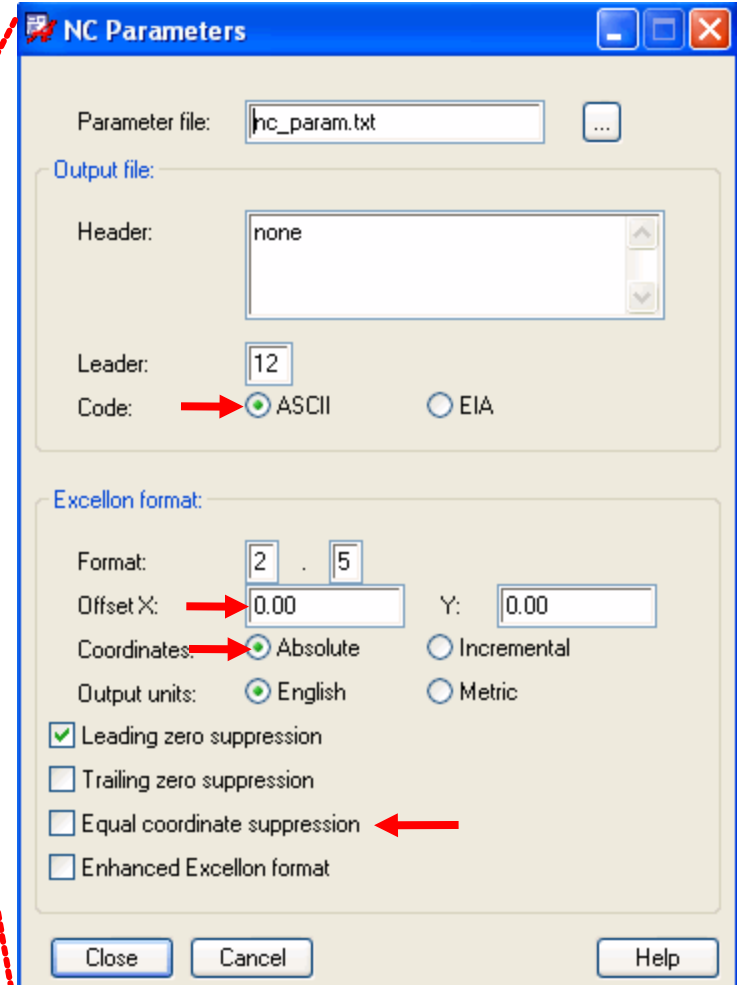
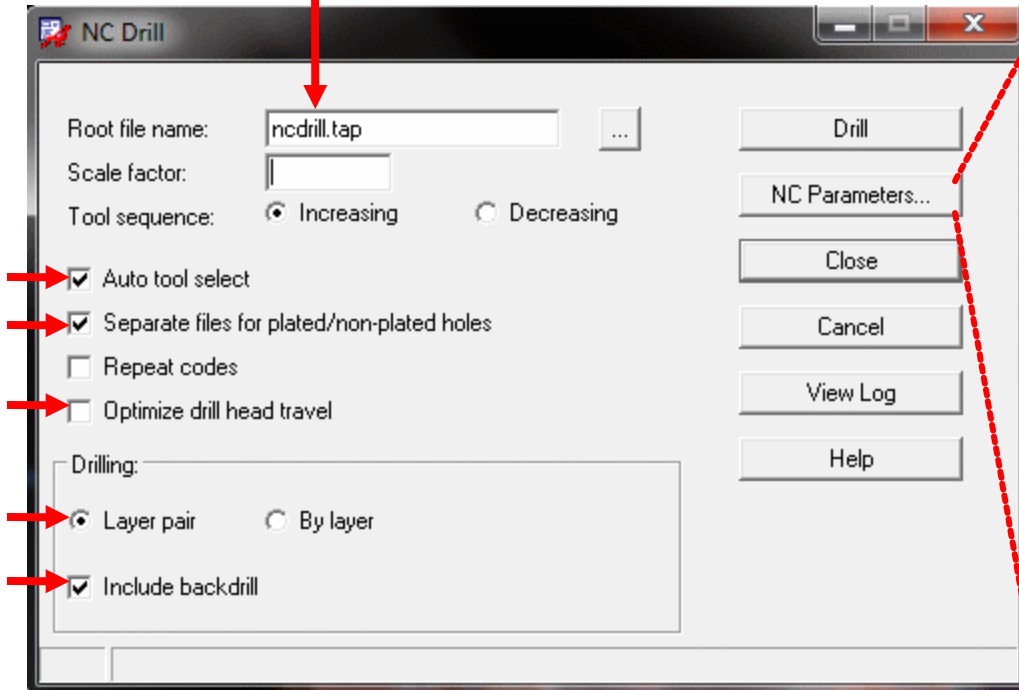
Preparing Cadence/Orcad Drill Data

NC Drill parameter setup is extremely important!

- Make sure the **Cadence/Orcad Manufacture > NC > NC Drill** dialog is setup...
 - Root Filename setting is defined as “ncdrill.tap”
 - Scale factor is empty
 - Auto Tool Select is ON
 - Separate File for Plated/Non-plated holes is ON
 - Optimize is OFF
 - Drilling is set to Layer Pair
 - Make sure the **Include BackDrill** box is checked **ON** if Back Drill checking is desired
- Make sure the **Cadence/Orcad Manufacture > NC > NC Parameters** dialog is setup...
 - Output File code is set to ASCII
 - Format values are the same as Gerber format values (prevents drill location round-off)
 - Offsets are set to “0”
 - Coordinates are Absolute
 - Suppress Equal Coordinates is OFF
- Make sure the **Cadence/Orcad Manufacture > NC > NC Route** dialog is setup...
 - “Separate files for plated / non-plated routing” must be checked **ON**

Preparing Cadence/Orcad Drill Data

Root Filename is very critical



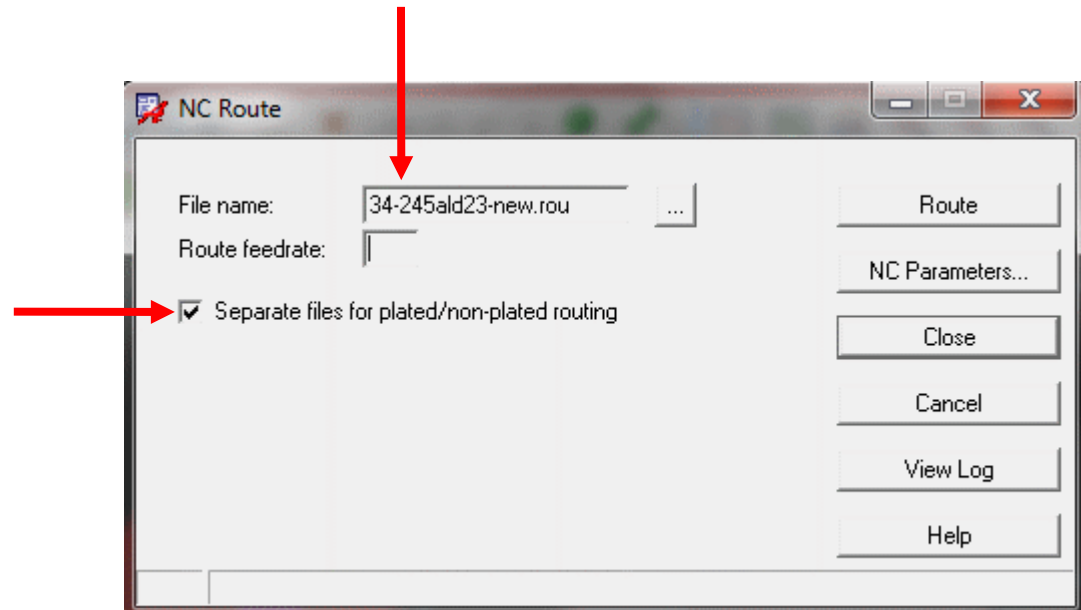
Arrows mark critical settings...

Preparing Cadence/Orcad Route Data

Make sure the **Cadence/Orcad Manufacture > NC > NC Route** dialog is setup...
“**Separate files for plated / non-plated routing**” must be checked **ON**

File name must be left 'as is' matching .brd filename

Make sure this is checked **ON** to be sure that plated and non-plated slots are separated within the Adiva database



Running the Cadence/Orcad Interface

- After an initial menu selection, no more than two to three button clicks are needed to extract Gerber & Drill, build the Adiva database and then run Netlist Compare providing connectivity analysis.
- Depending on board size and layer count, this process could take a few seconds to a few minutes to complete. Any wait time is solely dependant upon the size/qty of Gerber files created.
- Gerber and Drill files are created locally in the same directory as the .brd file (unless the *ads_sdart* variable is set) and also copied into a newly created directory containing all of the Adiva data. This directory is called “DRC_*jobname*” – where: “*jobname*” is the prefix to the .brd filename.
- Problems with data conversion are usually presented by an **alert!** posted to the graphics screen. If data conversion fails, first order of debug is to see if Allegro can create the data on its own through the Cadence/Orcad dialogs.
 - If it can, check artwork and drill dialog settings defined earlier. If they are correct, then Contact Adiva for further guidance.
 - If they are incorrect, make the appropriate setting change and try again.
- Basic flow of the interface is....

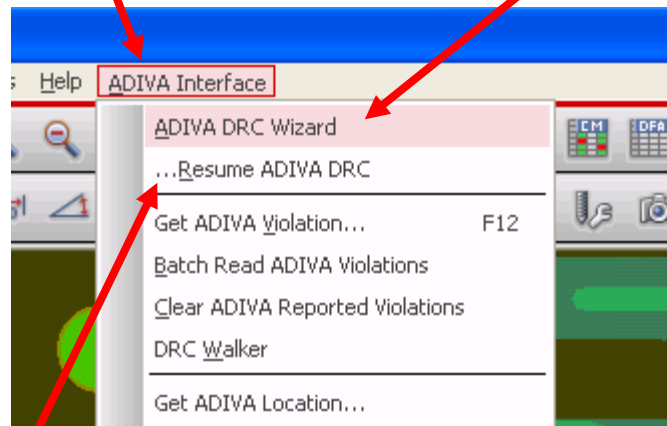
Start Interface

- >>> Select / Deselect / Adjust layers to convert Gerber to Adiva
- >>> Choose a DRC Rule file (if desired) and/or Custom DRC Checking file(s)
- >>> Adjust aperture settings for Gerber 6x00 data (only if standard Gerber is used for output)
- >>> Watch Adiva appear on screen and produce Net Compare results when conversion completes.

Running the Cadence/Orcad Interface

Cadence/Orcad Menu Selection

Choose the ADIVA DRC Wizard



The “**...Resume ADIVA DRC**” menu selection opens an existing Adiva database that may have been created on a previous occasion – there is no need to rebuild the database if it already exists.

Running the Cadence/Orcad Interface

Listing of artwork as defined in Cadence/Orcad. Un-check a particular Artwork listing to prevent creation and addition to the Adiva database.

(Example: un-check assy drawings and drill drawings as they are not needed for artwork analysis)

Select and Load Master DRC Rule File: the default setting which opens Adiva for graphical interaction after a user chooses to either load a Master Rule File or define rules within the DRC toolset.

Select and Run Custom DRC Commands:

A choice setting which opens Adiva for graphical or non-graphical processing of Custom DRC checks.

Either of these options can be selected singly or both can be processed.

• either or both of these can be programmed default on / off

When all definitions appear correct, select **Continue** to advance the conversion process

Include	Film Name	Layer Usage	Layer Number
<input checked="" type="checkbox"/>	TOP	Top Circuit	1
<input checked="" type="checkbox"/>	IN1	Inner Circuit	2
<input checked="" type="checkbox"/>	GND	Plane	3
<input checked="" type="checkbox"/>	VCC	Plane	4
<input checked="" type="checkbox"/>	IN2	Inner Circuit	5
<input checked="" type="checkbox"/>	BOTTOM	Bottom Circuit	6
<input checked="" type="checkbox"/>	silktop	Top Marking	51
<input checked="" type="checkbox"/>	silkbot	Bottom Marking	52
<input checked="" type="checkbox"/>	pastetop	Top Paste	53
<input checked="" type="checkbox"/>	pastebot	Bottom Paste	54
<input checked="" type="checkbox"/>	masktop	Top Mask	55
<input checked="" type="checkbox"/>	maskbot	Bottom Mask	56
<input checked="" type="checkbox"/>	outline	Outline	95
	Plated Thru Holes from TOP to BOTTOM		61
	Non Plated Holes from TOP to BOTTOM		62
	Buried Vias from TOP to IN1		63
	Buried Vias from IN2 to BOTTOM		64

Enable All Films Disable All Films

Select and Load Master DRC Rule File
 Select and Run Custom DRC Commands

Continue Cancel

Layers should be defined automatically – if any appear incorrect, just modify their layer type by menu selection. *

Layer numbers should also be set automatically. Layer numbers can be adjusted to user preference.

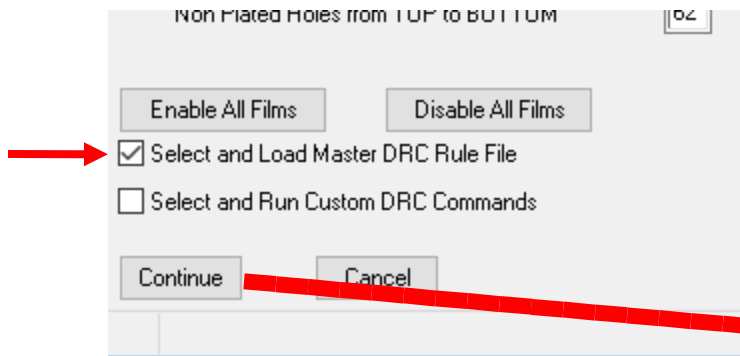
Layer numbers set as “-1” will not be converted into Adiva unless changed into a real positive layer number.

Holes, slots and buried or blind vias are automatically handled and mapped appropriately – user interaction is typically not required

* Layer types that are consistently listed incorrectly can be adjusted to eliminate user intervention. Contact Adiva for details.

Running the Cadence/Orcad Interface

In the Layer Assignment dialog, If you chose to **Select and Load Master DRC Rule File** then **Continue** as shown below...

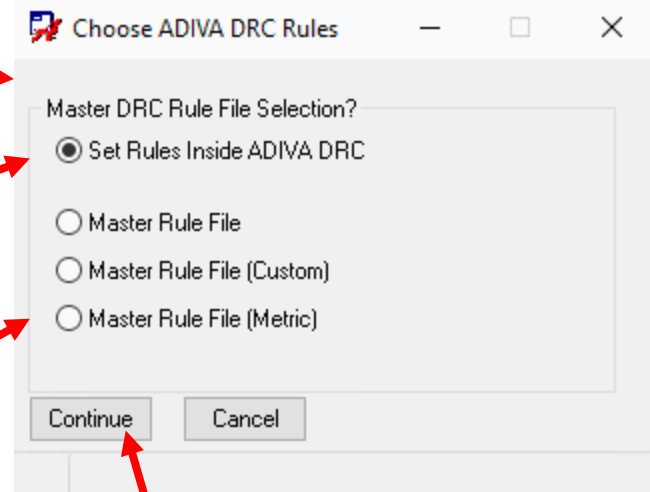


A new dialog will appear asking you to either - choose a predefined set of rules to load into the DRC tool when it opens -or- not load any predefined rules and **Set Rules Inside ADIVA DRC**.

This choice by default does not transfer any predefined rule sets allowing each ADIVA DRC rule menu to appear blank awaiting user interaction.

Predefined rules sets are listed for user choice. Only one rule set can be chosen.

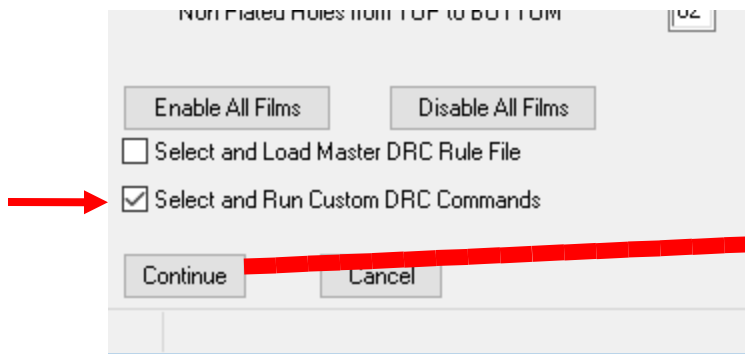
Contact Adiva directly for details on creating additional master rule files and including those files on this list for user choice.



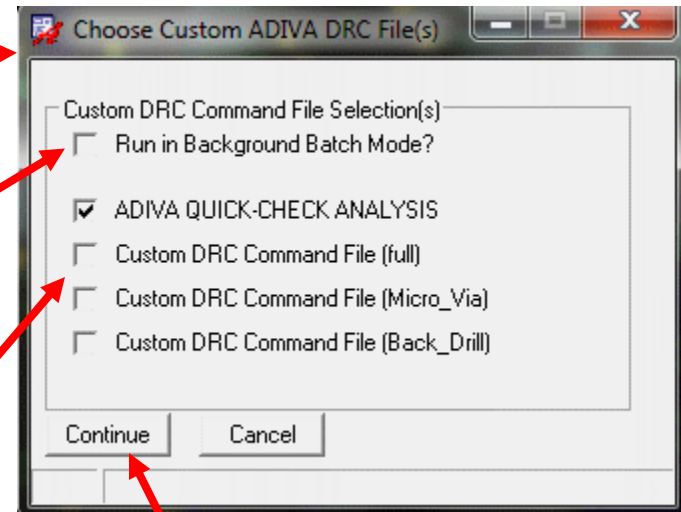
Select **Continue** to advance with the data conversion process loading any selected **Rule** file into the DRC dialogs.

Running the Cadence/Orcad Interface

In the Layer Assignment dialog, If you chose to **Select and Run Custom DRC Commands** then **Continue** as shown below...



A new dialog will appear asking you to choose a predefined set of **Custom DRC Checking Command file(s)**



Check this box ON to force Adiva's checking tools to operate in a background, "Black Box" mode. All Custom Command Files will process automatically without producing a graphical interface for user interaction. **this can be programmed to default ON*

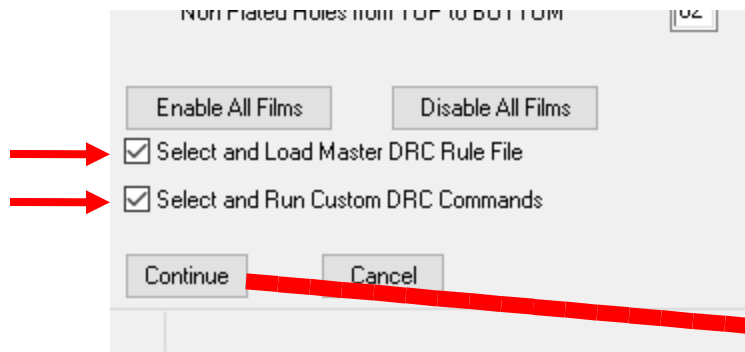
Predefined **Custom DRC Checking Command** files are listed for user choice. One or more **Custom Command Files** can be chosen.

Contact Adiva directly for details on creating additional **Custom DRC Checking Command** files and including those files on this list for user choice.

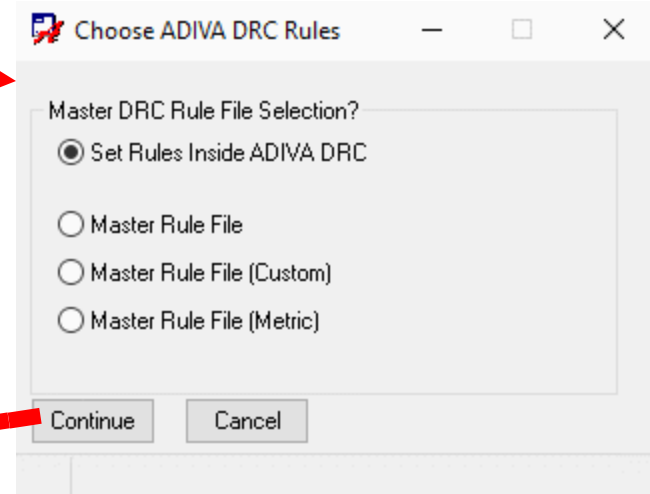
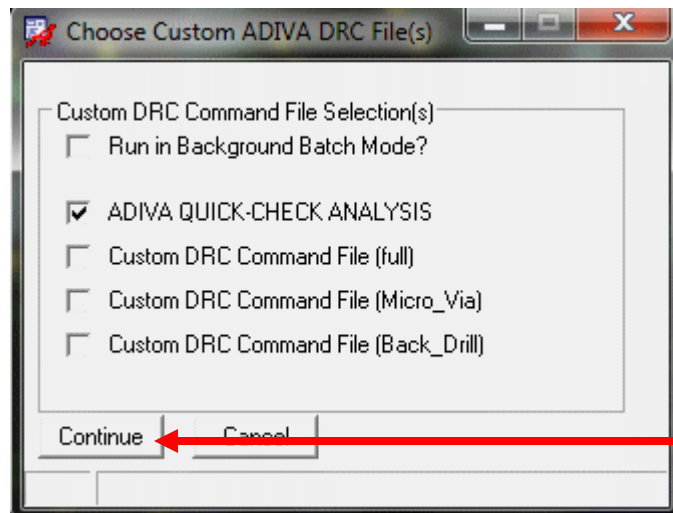
Select **Continue** to advance with the data conversion process executing all selected **Custom Command** files(s).

Running the Cadence/Orcad Interface

In the Layer Assignment dialog, If you chose both **Select and Load Master DRC Rule File** and **Select and Run Custom DRC Commands** then **Continue** as shown below...



Both previously shown **Rule** file and **Custom Command** file(s) selection dialogs will appear in the order depicted and function as previously described.



Select **Continue** to advance with the data conversion process. A master **Rule** file will load the DRC dialogs and **Custom Command** files will be automatically processed.

Running the Cadence/Orcad Interface

*** When converting 274X Gerber data – this dialog will not appear – skip this page

When converting Gerber 6x00 – this dialog may appear if certain FLASHED objects require further

definition not contained in the Gerber aperture listing (ie: thermals, special objects, etc)

Padstack name associated with Flash Name in Cadence/Orcad

To define aperture size and shape, select the Assist button

Choose a shape definition

Flash Name as listed in Cadence/Orcad photoplot.log file

Flash object definition in ADIVA aperture data format

Select **Continue** to advance the conversion process after dialog completion

Flash Name	Padstack	DRC Equivalent	Assist
DONUT-50	50D125NP	dn50x40	...
THRM-115	125R90	t115x90x15x1	...
THRM-145	125R90	t135x125x25x1	...
THRM-150	250R133	t310x300x59x1	...
THRM-175	150R133	t310x300x59x1	...
THRM-200	50D100NP	t60x50x10x1	...
THRM-40	VIA-12	t37x27x5x1	...
THRM-50	50S24	t60x50x10x1	...
THRM-60	50R24	t60x50x10x1	...
THRM-65	55R38	t65x45x10x45	...
THRM-70	62R43	t72x62x12x1	...

Stores definitions for use on future designs

Update Master Hints File

Continue Cancel

Map Aperture D37 DON...
Aperture Shape: Ring
c<outer>x<inner>
Outer Diameter: 50.00
Inner Diameter: 40.00
Update Cancel

Running the Cadence/Orcad Interface

When the conversion process completes, the **ADIVA DRC Analysis** tool will appear on the screen...

The screenshot displays the ADIVA DRC Analysis tool interface. The main window shows a green PCB layout with various components and traces. A white text box with a red border is overlaid on the layout, containing the text: "Wait for it to report Process Finished before proceeding". A red arrow points from this text box to the status window at the bottom of the interface. The status window displays the following text:

```
....Netting: ** Normal Termination **....Running Cad Netlist Compare using net.crf
....Cad Netlist Compare Done.
....Writing DRC_demo_e5.adi
....Writing Done
....Process Finished.
```

The interface also includes a menu bar (File, Edit, View, Window, Add, Special, Macros, Analysis, Toolkit, Help), a toolbar with various icons, and a right-hand panel with a legend for different PCB layers and components. The legend includes items such as "1 Top Circuit", "2 Inner Circuit", "3 Plane", "4 Plane", "5 Inner Circuit", "6 Bottom Circuit", "31 Top Paste", "32 Bottom Paste", "41 Top Mask", "42 Bottom Mask", "51 Top Marking", "52 Bottom Marking", "61 Plated Thru Holes", "62 Non Plated Holes", "63 Buried Vias", "64 Buried Vias", "90 Outline", "91 Top Cad", "92 Bottom Cad", and "93 Hole Cad".

Running the Cadence/Orcad Interface

Once the Adiva database has been created, displayed and the “**Process Finished**” message posted, there are several functions that can be performed:

CAD Netlist Compare has happened automatically. Select the **NetCmp** tab in the action display to view results of this function. See the **ADIVA Netlist Compare Guide** for further details.

AdivaView can read and display this ADIVA database. It is already saved and exists in the DRC_jobname directory created under the .brd file directory. Its filename format is “DRC_jobname.adi”
Where: “jobname” is the .brd file prefix and “.adi” is the ADIVA filename suffix.

DRC Analysis checks can be performed on this data looking for design rule violations. See the **ADIVA DRC Checking Guide** for further details.

Once the Adiva database is created, Cadence/Orcad can be closed if desired as there is no required connection needed for Adiva to function. One advantage however to leaving Cadence/Orcad open is for violation link-back of items found by Adiva into Cadence/Orcad. That way, a designer can repair a violation in real time. See the **ADIVA to Cadence/Orcad Violation Link-Back Guide** for further details.

This completes the **Cadence/Orcad to Adiva** Interface.

END

**Cadence/Orcad to Adiva
Interface
(Quick-Start User Guide)**