



PCB SOLUTIONS
An Orbotech Valor Company

GENESIS 2000



RELEASE NOTES VERSION 9.1D

Software Version 9.1d

Document 0991d0407
Published April 2007

© 2007 Frontline PCB Solutions LP
All rights reserved.

This document contains proprietary information belonging to Frontline PCB Solutions LP. This information is not for publication and is issued on condition that it will not be copied, printed or disclosed to a third party, without the written consent of Frontline PCB Solutions LP.

This document is subject to change without notice.

Frontline PCB Solutions LP
4 Faran St., Ashtrum Park
Northern Industrial Area,
P.O. Box 680 Yavne 81106 Israel
Tel: (972)-8-9322183
Fax: (972)-8-9322186
web page: <http://www.frontline-pcb.com>
email: info@frontline-pcb.com

Table of Contents

Chapter 1 Overview

Introduction	5
Identification	5
Intended Readers	5
Organization of this Manual	5

Chapter 2 Highlights

New Impedance Coupon Generator	6
Enhancements to Galvanic Etch Compensation DFM ..	6
Info Command for AOI Interface	6
User Restrictions for Running and Debugging Scripts ..	6

Chapter 3 Major Enhancements

Enhancements to Galvanic Etch Compensation DFM ..	7
New Action Screen Parameters	7
Enlarge Gold Pads By	7
New Line Mode Commands for AOI Interface	8
Info Command for AOI Interface	9
User Restrictions for Running and Debugging Scripts	11
Notes regarding Actions/Script Action	11
Enhancements to Dynamic Etch Compensation	12
New ERF Variables control priority assignments	12
New System Attribute to Indicate Smoothing Lines ...	12

Chapter 4 Impedance Coupon Generator

Terminology	13
Activation	16
Workflow	17
Manual Workflow	17
Stage 1 - Table of Constraints	17
Stage 2 - Grouping	21
Stage 3 - Coupon Parameters	23
Automatic Workflow	25
Stage1: Table of Constraints	25
Stage 2: Grouping	25
Stage 3: Coupon Parameters	26
Stage 4: Return to Table of Constraints	26

Stage 5: Grouping Parameters	26
Configuration	27
General Configuration Parameters	27
Text Parameters	30
Test Models	31
Test Model Parameters	31
Valid Test Models	33
Layouts	34
What is a valid layout?	34
Layout Parameters	34
Coupons	36
What is a valid coupon?	36
Coupon Parameters	37
Impedance Coupon Generator - Layout Types	38
Layouts with mixed rows	38
TUTORIAL: How to Draw a Constraint into an Impedance Coupon	42
TUTORIAL: Transfer of Constraints to Impedance Coupon	45
Line Mode Commands	47

Chapter 5 Parameters, Attributes, and Commands

New Configuration Parameters	48
edt_auto_f_inf_window	48
iol_274x_ip_background	48
New Line Mode Commands for the Impedance Coupon Generator	49
impcpn_open	49
impcpn_close	49
impcpn_read_cfg	49
impcpn_edit_imptbl_start	49
impcpn_add_cnst	50
impcpn_edit_imptbl_end	50
impcpn_add_grp	51
impcpn_change_grp	51
impcpn_delete_grp	52
impcpn_set_cpn_prms	53
impcpn_generate_cpns	54
New System Attributes	54
New Line Mode Commands to Control Access to Scripts	54
script_debug_hide	54
script_debug_show	55
script_bind_show	55
script_bind_hide	55
script_run_show	55
script_run_hide	56
New Line Mode Commands for Orbotech AOI Interface	56
cdr_get_flow_stage_status	56
cdr_get_step	56

cdr_get_machine	56
cdr_get_nom_line	57
cdr_get_nom_space	57

Chapter 6 KIT List

Chapter 1 *Overview*

Introduction

This document describes the changes implemented in Genesis software version 9.1d as compared to the last major version 9.1c1.

Identification

The **get** process version number for all platforms is v9.1d

Intended Readers

All Genesis users.

Organization of this Manual

Chapter One, Overview - introduces the subject of release notes.

Chapter Two, Highlights - describes in brief the changes in this version.

Chapter Three, Major Enhancements - Lists major changes and enhancements made to Genesis in this version.

Chapter Four, Impedance Coupon Generator - Describes the new Impedance Coupon Generator. This new option generates impedance test coupons automatically, based on parameters you define at setup.

Chapter Five, Parameters, Attributes, and Commands - Lists changes and additions to configuration parameters, system attributes, and line mode commands.

Chapter Six, KIT List - Lists KIT bugs and requests that have been closed.

Chapter 2 *Highlights*

New Impedance Coupon Generator

The new Impedance Coupon Generator generates impedance test coupons automatically, based on parameters you define at setup.

See [“Impedance Coupon Generator” on page 13.](#)

Enhancements to Galvanic Etch Compensation DFM

New Screen Parameter **Enlarge Gold Pads By** enables the action to enlarge Gold Plated Pads.

See [“Enhancements to Galvanic Etch Compensation DFM” on page 7.](#)

Info Command for AOI Interface

The line mode command **info** is a powerful tool used to retrieve information from database entities to a file. This section describes a new **info** command made specifically for the Genesis to AOI Interface.

See [“Info Command for AOI Interface” on page 9.](#)

User Restrictions for Running and Debugging Scripts

Permission to run and debug arbitrary scripts may now be controlled using the `$GENESIS_DIR/share/privs` file. For details, see [“User Restrictions for Running and Debugging Scripts” on page 11.](#)

Chapter 3 *Major Enhancements*

Enhancements to Galvanic Etch Compensation DFM

New Action Screen Parameters

Enlarge Gold Pads By

This parameter enables you to define a function that will be used to calculate the enlargement of Gold Pads.

Default is **.zero** = no enlargements.

Note This enlargement is limited to working only in **Total** mode = **v_ratio_calc_mode = 1**.

Action Screen

Galvanic Etch Compensation ERF: First model (Mils) ?

Apply To :
☒ All ☐ Selected

Copper Area / Gold Area Max ratio : 25 % ☐ Ignore Ratio

Enlarge OSP Pads by : .zero

Enlarge Gold Pads By : .zero

Top Gold Mask Layer : 1

Bottom Gold Mask Layer : 1

Gold Plating Attribute : .gold_plating

Copper Spacing : 5 ml

Drill Spacing : 10 ml

Required SM Clearance AR : 2 ml

Required SM Coverage : 3 ml

Fix Spacing & SM Coverage By :
☒ Shave ☐ ReShape
 Work Mode :
☐ Report ☒ Repair

Consider Drilled OSP Pads Yes ☐

Create Report Layer : No ☐

Report Layer Suffix: report

If Report Layer Exists: Overwrite ☐

Ranges...

No results

Close

New Line Mode Commands for AOI Interface

cdr_set_flow_entry_status - The command is used for finding the red/green status of a flow stage.

cdr_get_step - The command is used for getting the name of the step loaded in the cdr interface

cdr_get_machine - The command is used for getting the name of the machine loaded in the cdr interface

cdr_get_nom_line - The command activates an analysis that calculates the layer's nominal line and returns it.

cdr_get_nom_space - The command activated an analysis that calculates the layer's nominal spacing and returns it.

For details, see [“New Line Mode Commands for Orbotech AOI Interface” on page 56](#).

Info Command for AOI Interface

The line mode command **info** is a powerful tool used to retrieve information from database entities to a file. **Info** works on entities in memory, thus its output represents the entity current data, even if the entity was not yet saved to disk.

This section describes a new **info** command made specifically for the Genesis to AOI Interface.

Entity name:\$JOB/\$STEP/\$LAYER/\$CDRSET

Entity: orbotech-aoiset

Options:

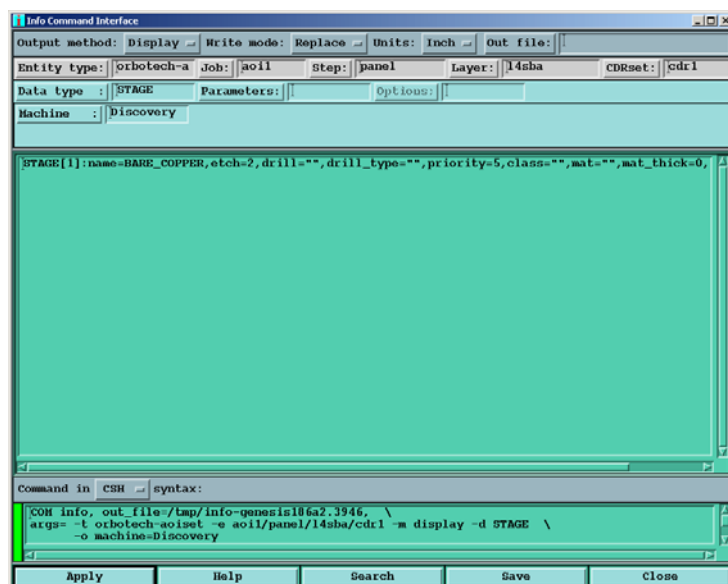
machine name of machine: vision, pc14, inspire, infinex, discovery
(Case insensitive)

stage name of stage from configuration file
(e.g. TIN_LEAD_BEFORE)

Data Type	Parameters	Description	Required options
STAGE		Information about stages (lists)	
	name		machine
	etch		machine
	drill		machine
	drill_type		machine
	priority		machine
	class		machine
	mat		machine
	mat_thick		machine
	cu_weight		machine
FTRS		Information about layer design	
	nom_line		
	min_line		
	nom_space		
	min_space		
	nom_ar	nominal annular ring	
	min_ar	minimal annular ring	
	nom_nfp_d	nominal nonfunctional pad spacing	
	min_nfp_d	minimal nonfunctional pad spacing	
	nom_plane_d	nominal place spacing	
PRMS		AOISET parameters	
	type		
	pattern		
	pol		

Data Type	Parameters	Description	Required options
	db		machine
	rule		machine
ALIGN		Alignment transformation	
	mirror		machine, stage
	rotate		machine, stage
	offset_x		machine, stage
	offset_y		machine, stage
ALIGN_TGT	x,y	List of alignment targets	stage
VRS_TGT	x,y	List of vrs targets	
DISH_DOWN_TGT	x,y	List of dish down targets	
REG_TGT	x,y	List of registration targets	
CALIB_TGT	x,y	List of calibration targets	
SCAN_ALIGN	x,y	List of scan alignment targets	
IMPED_TGT	x,y	List of impedance targets	
THICK_TGT	x,y	List of thickness targets	
INSPECT		Bounding box of inspection & exclusion area in panel coordinates	
	xmin		
	xmax		
	ymin		
	ymax		

Here you can see some of these new parameters in action in the Info Command Generator.



User Restrictions for Running and Debugging Scripts

Permission to run and debug arbitrary scripts may now be controlled using the `$GENESIS_DIR/share/privs` file. This file enables the system administrator to enable/disable certain line mode commands according to the privilege level of the user or according to their application user name.

The following new line mode commands are supported. For details, see [“New Line Mode Commands to Control Access to Scripts” on page 54](#).

script_debug_show - Shows the C-shell debugger

script_debug_hide - Hides the C-shell debugger

script_bind_show - Shows the script binding window

script_bind_hide - Hides the script binding window

script_run_show - Shows the script run window

script_run_hide - Hides the script run window

Adding one or more of these line mode commands to the `privs` file will enable or disable users from running, debugging, or binding scripts.

To control recording scripts, the line mode command **script_record** already exists.

Example

Add the following commands to the `privs` file to permit running, debugging, or binding scripts only to users with a privilege level of 95 or higher.

```
group scriptcontrol {
  global_priv 95
  users {
  }
  commands {
    script_run_show
    script_bind_show
    script_debug_show
  }
}
```

Notes regarding Actions/Script Action

- If a user does not have permission for the **script_run_show** line mode command, he cannot open the Script Action.
- If "Script Action" appears in a checklist, the user can change the parameter of the script path to run any script he wants from the checklist action. A "pre" line hook added to the line mode command **chklist_cupd**, which lists what scripts the user is allowed to run, could be used to deal with this problem.

Enhancements to Dynamic Etch Compensation

New ERF Variables control priority assignments

Four new ERF variables were added to the Dynamic Etch Compensation DFM action to assign priorities for etch compensation by feature type.

priority_pad = 0 # compensation priority for pads.
0 is highest priority

priority_smd = 0 # compensation priority for SMDs.
0 is highest priority

priority_trace = 0 # compensation priority for traces.
0 is highest priority

priority_other = 0 # compensation priority for others.
0 is highest priority

When there is not enough space to compensate both features, the feature with the higher priority receives the maximum possible compensation; the feature with the lower priority gets the rest.

Example

```
priority_pad = 2  
priority_smd = 1  
priority_trace = 0  
priority_other = 3
```

New System Attribute to Indicate Smoothing Lines

Smoothing lines are marked with the new system attribute **.detch_smooth**.

Chapter 4 *Impedance Coupon Generator*

The Impedance Coupon Generator (hereafter referred to as **ICG**) generates impedance test coupons automatically, based on parameters you define manually at setup.

Impedance test coupons are typically small PCBs with the same layer and trace construction as the main PCB. Impedance test coupons are fabricated at the same time as the source PCB so the coupon will have the same impedance as the PCB. The same apertures used for the controlled impedance traces are used to produce the test traces on the coupon.

The ICG receives a list of user-defined impedance constraints. It then builds one or more new impedance coupon step(s), and allocates impedance lines in an impedance coupon for each impedance constraint.

Terminology

Test model

Impedance test model

This version of the ICG supports two test models: **single ended** and **differential**. The test model definition includes the test model type and some additional parameters.

The available test models are defined in a configuration file. For more information, see [“Test Models” on page 31](#).

Impedance constraints

A set of parameters describing an impedance-controlled trace. Each impedance constraint definition includes the parameters listed below.

Test layer - Name of the layer that contains the impedance-controlled trace.

Test model - Name of impedance test model.

Top reference layer - Name of the top reference layer.

All layers above the test layer, up to the **top reference layer** (inclusive), influence the impedance of the trace.

Bottom reference layer - Name of the bottom reference layer.

All layers below the test layer, down to the **bottom reference layer** (inclusive), influence the impedance of the trace.

Artwork line widths - Line widths of the impedance line.

Required line widths - Line widths of the impedance line as specified by the circuit planner. (Optional parameter)

	<p>Artwork spacing - Space between two impedance lines that belong to the same constraint. This parameter is relevant for the Differential test model only.</p> <p>Required spacing - Space between two impedance lines that belong to the same constraint as specified by the circuit planner. This parameter is relevant for the Differential test model only. (Optional parameter)</p> <p>Impedance - The required impedance (in ohms).</p> <p>Tolerance - Impedance tolerance (in ohms).</p> <p>Group number - Number assigned to a defined cluster of impedance constraints. ICG builds one or more impedance coupon(s) for each defined group of constraints. See “Group (of constraints)” on page 14.</p>
<i>Span (impedance constraint)</i>	The list of layers that influence the impedance of the impedance-controlled trace. This includes all layers from the top reference layer (inclusive) to the bottom reference layer (inclusive) .
<i>Involved layers (impedance constraint)</i>	All layers in the span, except for the reference layers.
<i>Uninvolved layers (impedance constraint)</i>	All layers in the <i>matrix</i> , except for layers in the span.
<i>Foreign spans</i>	If two spans (layer lists) do not share any layer, they are considered as foreign spans.
<i>Reference Layer</i>	Acts as a barrier against influencing an impedance-controlled trace. Layers that lie above the top reference layer or below the bottom reference layer do not influence the impedance of the impedance-controlled trace.
<i>Consecutive constraints</i>	Two constraints are consecutive if the test layer of one constraint is part of the span of the second constraint.
<i>Group (of constraints)</i>	A defined cluster of impedance constraints allocated to the impedance test coupon. (See the first item in “Grouping” on page 14 .)
<i>Grouping</i>	<p>Each impedance constraint should be assigned to a specific group.</p> <p>A group is identified by its group number. A constraint is assigned to a specific group by defining its group number.</p> <p>During the process of generating impedance coupons, the ICG builds a unique impedance coupon for each group of constraints. That coupon will include all the impedance constraints that belong to the group.</p>

You should use more than one group of constraints if the coupon is not large enough to house all of the impedance constraints defined for the step.

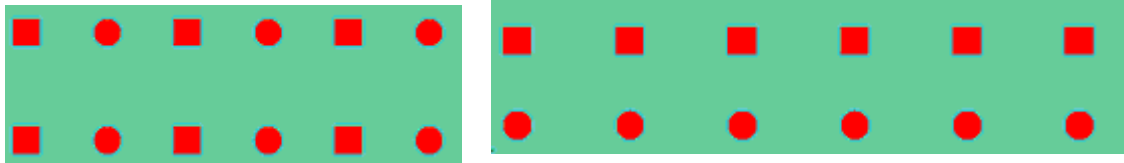
Signal test pad A test pad that is connected to the impedance line.

PG test pad A test pad that is connected to the reference layer(s) of the impedance line.

Coupon test pads layout Layout used to spread test pads over the coupon space.

Layout A set of parameters that defines how test pads should be spread over the coupon space. Available layouts are defined in the configuration file of the ICG. See [“Layouts” on page 34](#) for more information.

Two sample layouts are given below.



Channel A channel is formed by either:

- The coupon profile and a row of test pads.
- Two rows of test pads.

Coupon A set of parameters that defines coupons that can be used. Used primarily to define coupon size.

Available coupons are defined in the configuration file of the ICG. See [“Coupons” on page 36](#) for more information.

- Known Limitations:*
1. ICG allocates up to two impedance lines in each channel (in each layer).
 2. Grouping limitations - None

Activation

To open the ICG:

1. Open the Graphic Editor and select the source PCB. The source PCB is the PCB step where impedance-controlled traces are located.
2. Select **Actions > Impedance Coupon** to open the application.

Test Layer	Test Model	Top Ref Layer	Bot Ref Layer	Artwork Line	Required Line	Artwork Spacing	Required Spacing	Imp	Tol	Group No.
pg1	se	sig1	sig3	6	0	10	0	0	0	0
sig2	se	sig1	sig3	7	0	10	0	0	0	0
sig2_2	df	sig1	pg2	8	0	10	0	0	0	0
pg2	df	sig1	sig3	0	0	0	0	0	0	0
New	:									

Action Buttons

Press **Read configuration** to re-read the ICG configuration files. This button can be used in any stage and at any time.

Note The ICG requires that you define a separate configuration file to specify layout patterns for the coupon. For details, see [“Configuration” on page 27](#).

Grouping Rules - Define your grouping rules.

For manual mode, see [“How to use the generate grouping option:” on page 18](#).

For automatic mode, see [“Stage1: Table of Constraints” on page 25](#).

Clear Groups - Sets group number of all constraints to zero.

View Coupons - Press to view coupons generated by ICG.

Delete Coupons - Press to delete all coupons generated by ICG in the last session.

Navigation buttons

Press **Next** to go to the next stage. Use this button after setting the current stage parameters.

Press **Back** to go back to the previous stage. Use this button if you want to change the parameters of the previous stage.

Press **Close** to close the application and free the license.

You can move back and forth between the different stages in the ICG wizard. If necessary, after coupons have been generated, you can open the Graphic Editor and edit the generated coupons directly.

Note You can manually edit the ICG configuration files at any time, and reread them by pressing the **Read Configuration** button in the first stage of the ICG wizard. This enables you to use the same set of impedance constraints with different configuration parameters.

Workflow

The ICG is a wizard-like application. The Wizard goes through three stages:

Stage 1 - Define the table of (impedance) constraints

Stage 2 - Define group parameters

Stage 3 - Define coupon parameters

In each stage, the parameters are saved in the wizard only until one of the navigation buttons is pressed. Once you move to another stage, line mode commands are called and the job data changes.

Manual Workflow

Stage 1 - Table of Constraints

Test Layer	Test Model	Top Ref Layer	Bot Ref Layer	Artwork Line	Required Line	Artwork Spacing	Required Spacing	Imp	Tol	Group No.
pg1	se	sig1	sig3	6	0	10	0	0	0	0
sig2	se	sig1	sig3	7	0	10	0	0	0	0
sig2_2	df	sig1	pg2	8	0	10	0	0	0	0
pg2	df	sig1	sig3	0	0	0	0	0	0	0
New	:									

In this stage, you define all impedance constraints. After they have been defined, press **Next**.

If you want to add or delete a constraint, press **Back** until you get back to the stage showing the table of constraints.

How to add an impedance constraint:

1. Press **New** and select the test layer from the list presented.

2. Press **Test model** and select a test model. A list of available test models is read from the ICG configuration file.
3. Press **Top reference layer** and select the top reference layer. The layer list will show only layers that are above the test layer.
4. Press **Bottom reference layer** and select the bottom reference layer. The layer list will show only layers that are below the test layer.
5. Set the **Artwork Line** width in mils/microns.
6. If this is a **differential** constraint, set the artwork spacing.
7. Set the **Group Number**.

Note The rest of the parameters are optional, and are used only for setting the **.imp_info** attribute of the test pad.

How to delete an impedance constraint:

1. Go to the constraint you want to delete, and press **Test layer**.
2. Select **<Delete>**.
3. Constraint will be removed from the table of constraints.

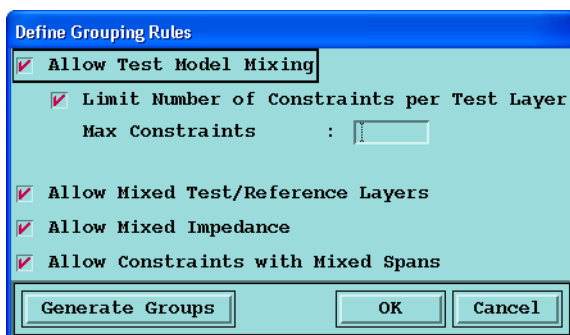
How to save the table of constraints to the job:

1. After all impedance constraints are defined, press **Next**. The job is updated, which means that the table of constraints of the current step is updated.
2. Press **File > Save** to save the modified job to the database.

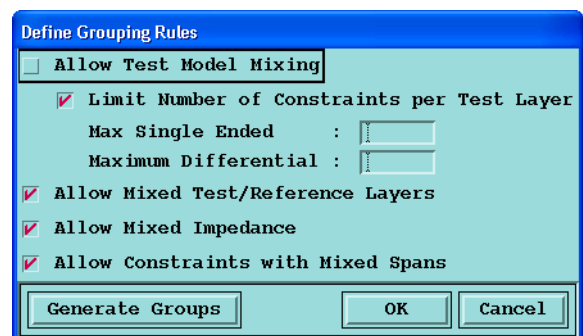
How to use the generate grouping option:

Note Before you define grouping rules, define the impedance constraints but do *not* define the constraint group number (leave it at zero).

1. In the Table of constraints, press **Grouping rules** to open the **Define Grouping Rules** popup.



Mixed test models (single-ended and differential) allowed.



Mixed test models (single-ended and differential) *not* allowed.

2. Define the grouping rules by clicking the desired check boxes.
3. Press **Apply**. The **Generate Groups** button becomes active.

4. Press **Generate Groups**.

The constraints in the table will be sorted into groups according to the rules you defined in this popup. The results will be displayed in the table of constraints. You can edit the results manually, if desired.

How the grouping rules interact with the defined impedance constraints is explained in the table below.

Table 1: How grouping rules interact with defined impedance constraints

Grouping Rules	Test model mixing		Mixed test/ref layers	Mixed impedance	Mixed span
	Limit number of constraints per test layer	Don't limit number of constraints per test layer			
Checked	Limit number of constraints per test layer	Don't limit number of constraints per test layer			
	A single group can include constraints of different test models. A single group will include no more than "Max constraints" per test layer	A single group can include constraints of different test models. A single group can include any number of constraints per test layer.	If a test layer with constraint A is the reference layer of constraint B, both layers can still belong to the same group.	A single group can include constraints with different impedance values.	If a test layer with constraint A is the reference layer of constraint B, both constraints can still belong to the same group.
Not checked	Limit number of constraints per test layer	Don't limit number of constraints per test layer			
	All constraints in the group must be of the same test model. A group with single ended constraints will include no more than "Max se constraint" per test layer. A group with differential constraints will include no more than "Max df constraint" per test layer.	All constraints in the group must be of the same test model. A group with single ended constraints can include any number of single ended constraints. A group with differential constraints can include any number of differential constraints.	If a test layer with constraint A is the reference layer of constraint B, the layers will be separated into two separate groups.	All constraints in the group must have the same impedance value.	If a test layer with constraint A is the reference layer of constraint B, both layers can still belong to the same group.

Stage 2 - Grouping

Groups are defined in the **Table of constraints** stage. In this stage you define the parameters of each group, and define the format used for the text that is added to every pad in the coupon.

After defining the parameters for all groups, press **Next**.

The screenshot shows the 'Impedance Coupon Generator' window at 'Stage 2 of 3: Group and Layout Parameters'. It features a 'View Coupons...' button and a 'Delete Coupons' button. Below these are input fields for 'Group Number' (1:2), 'Layout' (sig_pg_with_shift), and 'Coupon' (3x5). To the right, there are fields for 'Outer layers' and 'Inner layers', each with a 'Format' button. At the bottom is a table titled 'Defined Group Parameters'.

Group	Layout	Coupon	Size	Outer Lyr.	OL Format	Inner Lyr.
1	sig_pg_wit	3x5				
2	sig_pg_wit	3x5				

How to define the group parameters:

1. Press **Group Number** and select the group you want to edit.
2. Press **Layout** and select the required layout.
3. Press **Coupon** and select the required coupon.

(Option) How to define text assigned to pads in the Inner or Outer layers:

1. Press **Outer layers** and select the outer layer(s) to which you want to add text. Outer layers include the following layer types: silk screen, solder mask, solder paste, top and bottom copper layers.
2. Press the **Format** button to **Outer layer** and define the contents for the **Format** field. In each target layer, a text is added to every pad in the coupon. That text is defined by the contents of the Format field.
For details, see [“Text markers for pads” on page 22](#).
3. Repeat steps 1 and 2 for inner layers.

Once you have defined both the layout and the coupon, and (optionally) added text format parameters, the group is edited. You can view the group's parameters in the information display in the popup.

Impedance Coupon Generator
Stage 2 of 3: Group and Layout Parameters

View Coupons... Delete Coupons Read configuration

Group Number : 1
Layout : sig_pg_with_shift
Coupon : 3x5

Add text to the following layers:

Outer layers : sm-top Format : \$LYR
Inner layers : pg1 Format : \$REQ_SPACE

Group	Layout	Coupon : Size	Outer Lyr.	OL Format	Inner Lyr.	IL Format
1	sig_pg_wit	3x5	sm-top	\$LYR	pg1	\$REQ_SPACE
2	pg_sig_no_	3x5	sig1	\$LYR	sig2	\$REQ_SPACE

Close < Back Next > Generate Coupons >

Text markers for pads

In each target layer, a text will be added to every pad in the coupon. The text itself is defined by the format. The format can include the keywords listed below.

\$NLYR - Test or reference layer(s) number

\$IMP - Impedance

\$TOL - Tolerance for impedance

\$LYR - Test or reference layer/s name

\$MODEL - Test model name

\$REQ_LINE - Required line width

\$ART_LINE - Artwork line width

\$REQ_SPC - Required spacing

\$ART_SPC - Artwork spacing

Each *keyword* in the format will be replaced by a value according to the examples above; other characters will be copied as-is.

For example, text formatted as L\$NLYR will be replaced as follows:

For a signal test pad with an impedance line on layer5, L\$NLYR will be replaced by "L5".

For a P&G test pad with reference layers layer1 and layer8, L\$NLYR will be replaced by "L1/8".

Placement of pad text markers -

The text features will be calculated by ICG according to text configuration parameters. For details, see [“Configuration” on page 27](#).

Text for the bottom row is positioned under the test pads. For other rows, the text is positioned above the test pad.

Text is placed at a defined distance from the center of a pad. This distance is defined with the formula **pad_size/2 + the minimal required distance**

between pad and text. (This distance is defined in the configuration file as MIN_DIST_TEXT_2TEST_PAD.)

Stage 3 - Coupon Parameters

In this stage you define various parameters relating to coupon generation.

1. Define the coupon drill layer.

- To use an existing layer, press **Drill: Layer** and select the required drill layer.
- To create a new drill layer, type the layer name in the **Drill: Layer** field.

All coupon drills will be added to this drill layer.

2. Define drill size (Optional).

Press **Drill: Size** to call up a list of available drill sizes in the drill layer you selected previously. Select the required drill size.



Note If you do not set the drill size, drill size will be taken from the layout as defined in the configuration file.

3. Check **Add tooling holes** if you want to add tooling holes to the coupon. See “(Optional) Add Tooling Holes.” on page 24 for details.
4. Define the manufacturing limitations **Min. line widths** and **Min. spacing**. The application will check that all constraints meet these limitations.
5. Define the **Prefix** and **Suffix** for the coupon steps that will be created by the wizard. The coupon name will be:
suffix_<source step name>_<group number>_prefix.
6. Define the required shielding type: **Plane** or **Dotted**. If you select Dotted, the **Symbol**, **Dx** and **Dy** buttons become active. Press **Symbol** to open the **Symbol Selection Popup**. Use this popup to define the required symbol.
7. Select the action to occur when group constraints do not fit inside a single coupon.

8. (Optional) Select a coupon shrinking method by checking the **Shrink** button. Shrinking will be done according to the method you define.
 - a. Choose **Predefined** to have the coupon shrunk to a size matching one of the coupons defined in your configuration file. Press **Coupon Size** to call up a list of all available coupon sizes, and choose the required dimensions.

Note If you don't define a specific list of coupons, the coupon will be shrunk to one of the sizes defined in the configuration file.

- b. Choose **Round to** shrink the coupon to any size. Select the desired rounding method: whole, half, or quarter.

If you choose to add tooling holes, they will be added to the coupon according to the procedure detailed below.

9. (Optional) **Add Tooling Holes.**

If you choose to add tooling holes, they will be added to the coupon according to the procedure detailed below.

- a. When you check **Add tooling holes**, two tooling holes are added to the coupon.
 - b. The tooling holes are positioned on the right and left sides of the coupon, at a given distance from the coupon profile. This distance is defined in the configuration file as DIST_2TOOLING_HOLES.
 - c. The tooling holes are added to the coupon drill layer. This layer is defined in the Coupon Parameters stage of the ICG wizard. For details, see [“Define the coupon drill layer.” on page 23](#).
 - d. The tooling holes size is defined in the configuration file as TOOLING_HOLE_SIZE.

10. Click **Generate Coupons** to generate the coupons.

Automatic Workflow

In manual workflow, you define the impedance constraints and the groups. In the automatic workflow, you define the impedance constraints but *not* the group numbers. Groups are defined automatically, taking into account grouping rules (if you have defined them).

Stage1: Table of Constraints

1. Define impedance constraints, but do not define their group number.
2. Define grouping rules (Optional). See [“How to use the generate grouping option:” on page 18](#) for details.

Note If you define grouping rules, the automatic grouping process will take into account the grouping rules you have defined.

3. Press **Next**.

The screenshot shows the 'Impedance Coupon Generator' dialog box, Stage 1 of 3: Impedance Constraints. It features a table with columns: Test Layer, Test Model, Top Ref Layer, Bot Ref Layer, Artwork Line, Required Line, Artwork Spacing, Required Spacing, Imp, Tol, and Group No. The table contains four rows of data and a 'New' row. Below the table are buttons for 'Close', '< Back', 'Next >', and 'Generate Coupons >'. At the top, there are buttons for 'Grouping Rules...', 'Generate Groups', 'Clear Groups', 'View Coupons...', 'Delete Coupons', and 'Read configuration'.

Test Layer	Test Model	Top Ref Layer	Bot Ref Layer	Artwork Line	Required Line	Artwork Spacing	Required Spacing	Imp	Tol	Group No.
pg1	se	sig1	sig3	6	0	10	0	0	0	0
sig2	se	sig1	sig3	7	0	10	0	0	0	0
sig2_2	df	sig1	pg2	8	0	10	0	0	0	0
pg2	df	sig1	sig3	0	0	0	0	0	0	0
New	:									

Stage 2: Grouping

1. Define the **Layout** and **Max Coupon** parameters, and the **text** parameters.

Max Coupon defines the maximum coupon size, and all coupon parameters. All coupons will be generated with the selected layout. Coupon size will be similar to Max Coupon or smaller.

2. Press **Next**.

The screenshot shows the 'Impedance Coupon Generator' dialog box, Stage 2 of 3: Group and Layout Parameters. It includes fields for 'Layout', 'Max Coupon', 'Outer layers', 'Inner layers', and 'Format'. Below these fields is a table with columns: Group, Layout, Coupon, Size, Outer Lyr., OL Format, Inner Lyr., and IL Format. The table contains one row with the value 'All' in the 'Group' column. At the top, there are buttons for 'View Coupons...', 'Delete Coupons', and 'Read configuration'.

Group	Layout	Coupon	Size	Outer Lyr.	OL Format	Inner Lyr.	IL Format
All							

Stage 3: Coupon Parameters

1. Define coupon parameters. This is similar to the parameters you define in the manual work flow. (For details, see [“Stage 3 - Coupon Parameters” on page 23.](#))
2. Press **Generate Coupons** to create the coupon steps. The name of the new impedance coupon step will be **<name of source step>_impcpn_<group number>**.

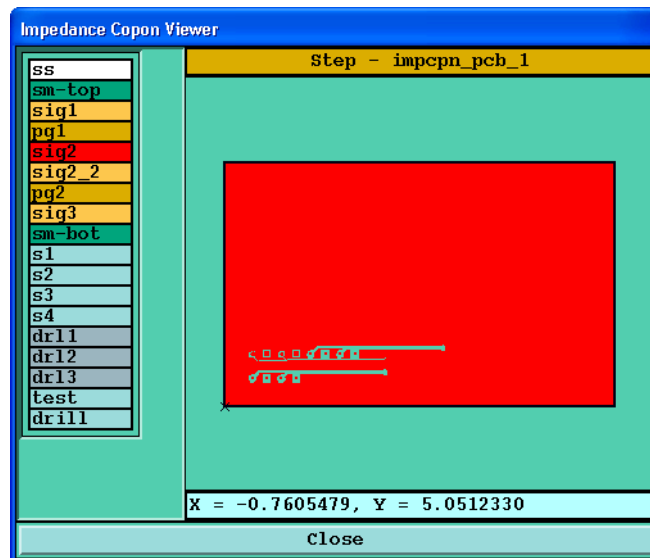
Stage 4: Return to Table of Constraints

After the coupons are created, the wizard will return to the first stage Table of Constraints where you will be able to see the auto-grouping results. Each constraint in the table now has a group number (not a zero) in the **Group No.** column.

Stage 5: Grouping Parameters

1. Press **Next** to see the list of groups created by the automatic grouping process and the groups' parameters, such as layout, coupon, and actual coupon size.
2. Press **View Coupons** to view the resulting coupons. The coupons can be viewed either in the Graphic Editor or in a separate viewer. This is defined by how you set the Genesis configuration parameter **imp_coupon_viewer**.

If **imp_coupon_viewer = internal viewer**, pressing **View** opens the coupon in a viewer.



If **imp_coupon_viewer = graphic editor**, pressing **View** opens the coupon in a graphic editor

3. You can now accept the results of the auto-configuration process as displayed, or return to the first stage of the wizard (Table of Constraints) to make any necessary modifications, and then regenerate the coupons.

On-Screen display

You can open impedance coupons in the Graphic Editor.

Each test pad added to the test layer in the impedance coupon is assigned the system attribute **.imp_info**. The attribute describes the impedance constraints for the source step. To see the values defined for this system attribute, follow the instructions below.

To view the test pad attribute if Genesis configuration parameter `edt_auto_f_inf_window = yes`.

1. Highlight the required test pad.
2. The Feature Information Popup opens. Here you can see all feature attributes and their values.

To view the test pad attribute if Genesis configuration parameter `edt_auto_f_inf_window = no`.

1. Highlight the required test pad.
2. Press the Shift key and hold it down.
3. Position the mouse on the information bar (bottom part of the editor).
4. Click **M1** and release the Shift key. The Feature Information Popup opens. Here you can see all feature attributes and their values.

Configuration

You must define the ICG configuration file in order to use the application.

Location of the configuration files:

HOME/.genesis/hooks/impedance_config (First priority)

<GENESIS_DIR>/hooks/impedance_config (Second priority)

The first line of the configuration file must be:

UNITS = INCH #Options: Inch/MM

The configuration file consists of four parts:

1. General parameters
2. Test models
3. Layouts
4. Coupons

General Configuration Parameters

UNITS

Options: INCH, MM

The units of measurement in which the file is written.

If **INCH**, all unit-sensitive parameters are in INCH except for aperture names - they are in mils.

If **MM**, all unit-sensitive parameters are in INCH except for aperture names - they are in microns.

ROUT_BELOW = yes

Options: yes, no

If set to yes, ICG will use the channel between the step profile and the first row of pads.

ROUT_BELOW_PRIORITY

Options: 1, 2, 3 (3 is highest priority)

Priority for routing impedance lines below first row of pads.

ROUT_ABOVE = yes

Options: yes, no

If set to yes ICG will use the channel between the step profile and the last row of pads.

ROUT_ABOVE_PRIORITY

Options: 1, 2, 3 (3 is highest priority)

Priority for routing impedance lines above last row of pads.

ROUT_BETWEEN = yes

Options: yes, no

If set to yes ICG will use the channel between every two rows of pads.

ROUT_BETWEEN_PRIORITY

Options: 1, 2, 3 (3 is highest priority)

Priority for routing impedance lines between rows of pads.

ALLOW_TWO_IMP_LINE_IN_ONE_CHANNEL = yes

Options: yes, no

If set to yes ICG will try to allocate up to two single-ended impedance lines in one channel.

ALLOW_ONLY_PARALLEL_DF_IN_MID_CHANNEL

Options: yes, no

If set to yes, a differential constraint in the middle of a channel can be allocated using two pads that are on the same x axis.

If yes, all differential impedance lines will look like this:



If no, some of the differential impedance lines may look like this:



Text Parameters

Define parameters for text additions to pads

Example

```
TEXT_PARAMS {
    MIN_DIST_TEXT_2TEST_PAD
    TEXT_POLARITY
    ADD_BACKGROUND
    BACKGROUND_MARGIN
    X_SIZE
    Y_SIZE
    LINE_WIDTH
    ALLOW_TEXT_MIRROR
}
```

MIN_DIST_TEXT_2TEST_PAD

Minimal required distance between a test pad and its text/labeling.

TEXT_POLARITY

Options: pos, neg, like_layer, opp_to_lyr

pos - text polarity is always positive.

neg - text polarity is always negative.

like_layer - text polarity is always similar to the layer polarity.

opp_to_layer - text polarity is always opposite to the layer polarity.

Note Defined for copper layers only. Text polarity on mask layers is always the same as layer polarity.

ADD_BACKGROUND

Options: yes, no.

If set to yes, ICG adds background under the text. Background polarity is always opposite the text polarity.

Note These parameters affect the text addition for signal, power ground and mixed layers only.

BACKGROUND_MARGIN

Margin for background dimensions calculation.

X_SIZE, Y_SIZE, LINE_WIDTH

Text size parameters.

If Y_SIZE is not defined, ICG will define y_size = pad size.

If X_SIZE is not defined, ICG will define x_size = y_size*5/7.

If LINE_WIDTH is not defined, ICG will define
line_width = y_size/7.

ALLOW_TEXT_MIRROR

Options: yes, no

If set to yes, the text in the bottom copper layer and in all mask layers (e.g solder mask, solder paste and silk screen) beneath it will be mirrored.

Test Models

Define all available test models.

Example

```
TEST_MODELS {
  se {
    TEST_MODEL_TYPE = SE #SE/DF
    DIST_PAD2CPR = 0.01 #pad2line & pad2shield
    OTHER_LAYER_OFFSET = 0.005
    DIST_LINE2CPR = 0.02 #line2line & line2shield
    ADD_AOI_PADS = yes
    AOI_PAD_DX = 0.05
    AOI_PAD_DY = 0.015
    IMP_LINE_LENGTH = 2
  }
  df {
    TEST_MODEL_TYPE = DF
    DIST_PAD2CPR = 0.025 #pad2line & pad2shield
    OTHER_LAYER_OFFSET = 0.01
    DIST_LINE2CPR = 0.02 #line2line & line2shield
    ADD_AOI_PADS = yes
    AOI_PAD_DX = 0
    AOI_PAD_DY = 0.010
    IMP_LINE_LENGTH = 2
  }
}
```

Test Model Parameters

TEST_MODEL_TYPE - Test model types

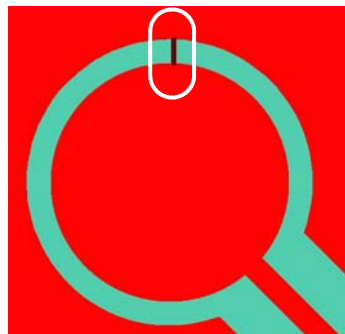
Options: SE (for single ended), DF (for differential)

SE - One impedance line

DF - Two impedance lines at a certain distance from each other

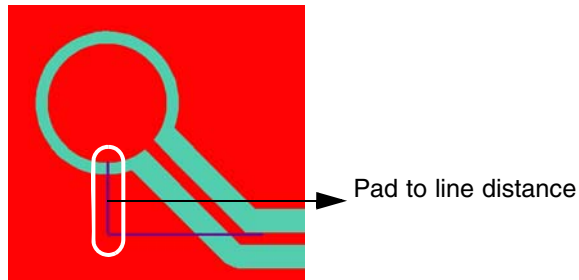
DIST_PAD2CPR - The minimum required distance between:

- Test pad and Impedance line
- Test pad and shielding: applicable only in test layer.

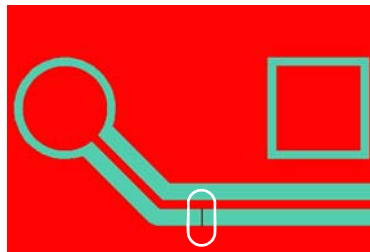


DIST_LINE2CPR - The minimal required distance between:

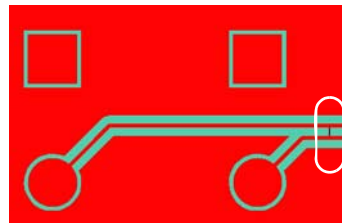
- Impedance line and test pad



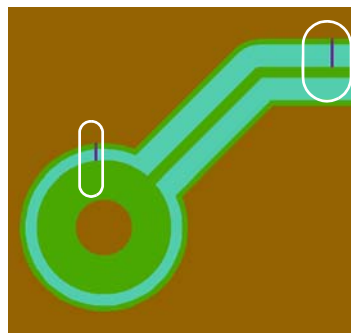
- Impedance line and shielding



- Single ended impedance lines that belong to different constraints. Applicable only in test layer.



OTHER_LAYER_OFFSET - Used to calculate DIST_PAD2CPR and DIST_LINE2CPR for the layers in the constraint span (layers other than the test layer and the reference layers). This parameter defines the additional spacing required for other layers in the span.



ADD_AOI_PADS

Options - yes, no

If set to yes, a pad is added at the end of the impedance line.

AOI_PAD_DX

Define X distance from impedance line.

Can be zero.

AOI_PAD_DY

Define Y distance from the end of the impedance line.

Can be zero.

Valid Test Models**Valid SE test model -**

$\text{Max}(\text{DIST_PAD2CPR}, \text{DIST_LINE2CPR}) > \text{AOI_PAD_DY}$

Otherwise the AOI pad may touch a test pad.



Single ended impedance line, with AOI pad in a
 $\text{aoi_pad_dx} > 0$ & $\text{aoi_pad_dy} > 0$

Valid DF test model -

$\text{AOI_PAD_DY} > 0$

Otherwise the AOI pads of the two impedance lines may touch each other.



Differential impedance lines, with AOI pad in
 $\text{aoi_pad_dx} > 0$ & $\text{aoi_pad_dy} > 0$

Layouts

Layout definition defines how to place the test pads over the coupon layers.

What is a valid layout?

A valid layout defines at least one signal pad and one PG pad, and defines a PITCH distance between a signal pad and a PG pad. Without these layout definitions, the probes can't be positioned on the Signal and PG pad at the same time.

Example

```
LAYOUTS {
  h_sig_pg  }
    FIRST_ROW_START_TEST_POINT = SIG
    FIRST_ROW_NEXT_TEST_POINT = PG
    SECOND_ROW_START_TEST_POINT = SIG
    SECOND_ROW_NEXT_TEST_POINT = PG
    SHIFT_ROWS = no
    ROWS_DY = 0.4
    ROW_PAD2PAD_DX = 0.3
    MAX_ROW_NUM = ANY
    MAX_COL_NUM = 20
    PITCH = 0.3
    TEST_LYR_SIG_APR = r100
    TEST_LYR_PG_APR = s100
    REF_LYR_SIG_APR = r100
    REF_LYR_PG_APR = thr150x100x0x4x20
    SM_LYR_SIG_APR = r100
    SM_LYR_PG_APR = s100
    DRILL_SIZE = 0.02
    AOI_PAD_SIZE = 0.02
    NFP_SIZE = 0.05
  }
}
```

Layout Parameters

FIRST_ROW_START_TEST_POINT

Options: SIG, PG

The type of the first pad in the first row of pads.

Also the type of every odd pad in every odd row of pads.

FIRST_ROW_NEXT_TEST_POINT

Options: SIG, PG

The type of the second pad in the first row of pads.

Also the type of every even pad in every odd row of pads.

SECOND_ROW_START_TEST_POINT

Options: SIG, PG

The type of the first pad in the second row of pads.

Also the type of every odd pad in every even row of pads.

SECOND_ROW_NEXT_TEST_POINT

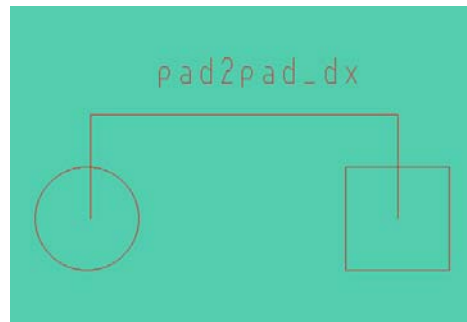
Options: SIG, PG

The type of the first pad in the second row of pads.

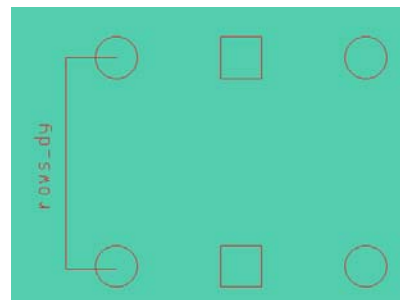
Also the type of every odd pad in every even row of pads.

ROW_PAD2PAD_DX

Distance between two pads in the same row of pads. Distance is measured center to center.

**ROWS_DY**

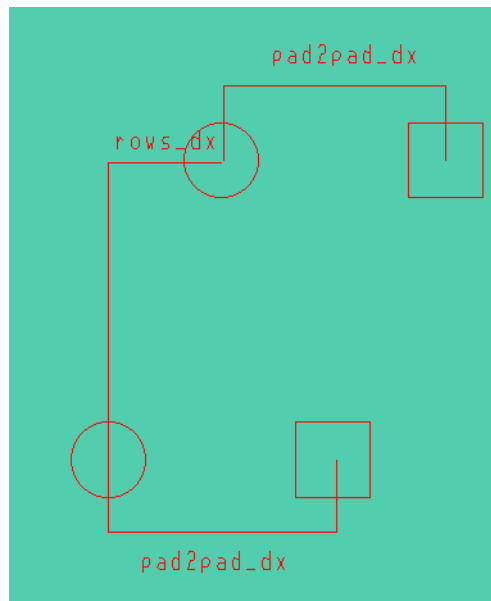
Dy between first pad in odd row of pads and first pad in an even row of pads. Distance is measured center to center.

**SHIFT_ROWS, ROWS_DX**

If set to **no**, the first pad in all rows is positioned in the same x coordinate.

If set to **yes**, the first pad in all odd rows has the same x coordinate, and the first pad in every even row has the same x coordinate.

The X coordinate of the first pad in the even rows is equal to the X coordinate of the first pad in the odd rows + ROWS_DX.

**MAX_ROW_NUM**

Options: number of rows or ANY

Maximum number of rows of pads.

If set to **ANY**, system will calculate the maximum number of rows taking into account coupon size, required distance, and other relevant parameters.

MAX_COL_NUM

Options: number of columns or ANY.

Maximum number of pads in each row of pads.

If set to **ANY**, system will calculate the maximum number of pads taking into account coupon size, required distance, and other relevant parameters.

PITCH

The distance between the probes of the impedance measurement equipment.

ICG expects the distance between every pair of test points (a pair being one PG test point and one SIG test point) to be equal to **PITCH** as defined in the configuration file.

Coupons**What is a valid coupon?**

In a valid coupon, **dist_profile2pad > dist_profile2line**. Otherwise, you cannot use the bottom and top channels.

Example

```
COUPONS {
  3x5 {
    COUPON_SIZE = 3X5
    DIST_PROFILE_2PAD = 0.5
    DIST_PROFILE_2LINE = 0.05
    ADD_LINE_AROUND_COUPON_PROFILE = yes
    PROFILE_LINE_WIDTHS = 0.01
  }
}
```

Coupon Parameters**COUPON_SIZE**

Format - mxn or mXn where m is the x dimension of the coupon and n is the y dimension of the coupon

DIST_PROFILE_2PAD

Minimal required distance between profile and test pad.

DIST_PROFILE2LINE

Minimal required distance between profile and impedance line.

ADD_LINE_AROUND_COUPON_PROFILE

Options: no/yes

If set to yes, 4 lines around the coupon profile will be added to each layer with a dotted pattern fill.

PROFILE_LINE_WIDTHS

If ADD_LINE_AROUND_COUPON_PROFILE = yes, this value sets the widths of the bounding lines.

MAX_SHIELD_SLIVER_SIZE

Used to remove all slivers less than the value defined in this parameter.

Note If this parameter is set and is greater than zero, then small slivers of the shielding contour will be removed.

MIN_SHIELD_CONT_SIZE

Used to remove all shield islands and holes less than defined value.

Note If this parameter is set and is greater than zero, then small holes and islands of the shielding contour will be removed.

DIST_2TOOLING_HOLES

If you add tooling holes to the coupon, this parameter defines the distance between the coupon profile and the tooling holes.

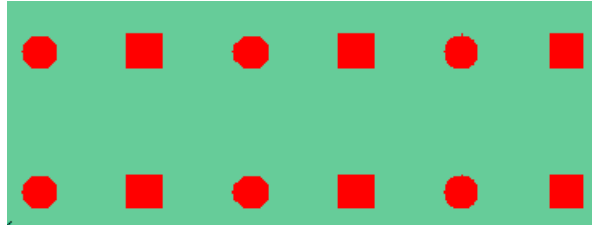
TOOLING_HOLE_SIZE

If you add tooling holes to the coupon, this parameter defines the size of the tooling holes.

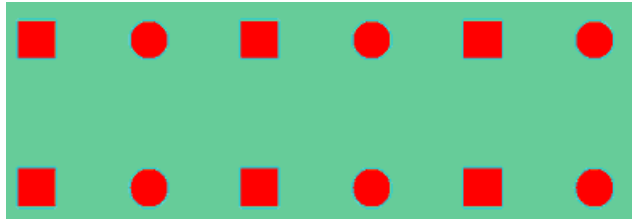
Impedance Coupon Generator - Layout Types

Following are images of all supported layouts and how each layout should be defined in the configuration file.

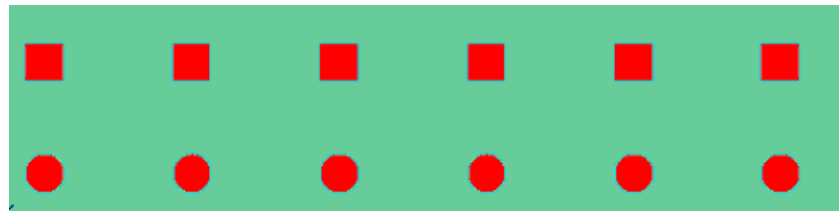
Layouts with mixed rows



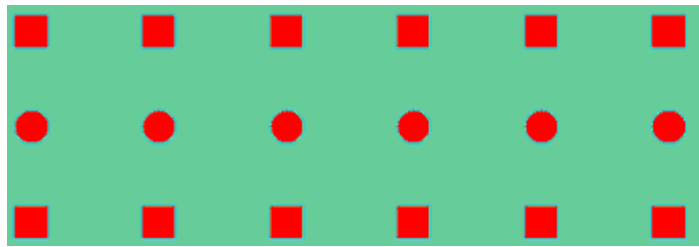
```
sig_pg_no_shift  }
    FIRST_ROW_START_TEST_POINT = SIG
    FIRST_ROW_NEXT_TEST_POINT = PG
    SECOND_ROW_START_TEST_POINT = SIG
    SECOND_ROW_NEXT_TEST_POINT = PG
    SHIFT_ROWS = no
    ROWS_DY = 0.4
    MAX_ROW_NUM = 2
    MAX_COL_NUM = ANY
    PITCH = 0.3
    TEST_LYR_SIG_APR = r100
    TEST_LYR_PG_APR = s100
    REF_LYR_SIG_APR = r100
    REF_LYR_PG_APR = thr150x100x0x4x20
    SM_LYR_SIG_APR = r120
    SM_LYR_PG_APR = s120
    DRILL_SIZE = 0.02
    AOI_PAD_SIZE = 0.02
    NFP_SIZE = 0.05
}
```



```
pg_sig_no_shift {  
    FIRST_ROW_START_TEST_POINT = PG  
    FIRST_ROW_NEXT_TEST_POINT = SIG  
    SECOND_ROW_START_TEST_POINT = PG  
    SECOND_ROW_NEXT_TEST_POINT = SIG  
    SHIFT_ROWS = no  
    ROWS_DY = 0.4  
    MAX_ROW_NUM = 2  
    MAX_COL_NUM = 6  
    PITCH = 0.3  
    TEST_LYR_SIG_APR = r100  
    TEST_LYR_PG_APR = s100  
    REF_LYR_SIG_APR = r100  
    REF_LYR_PG_APR = thr150x100x0x4x20  
    SM_LYR_SIG_APR = r120  
    SM_LYR_PG_APR = s120  
    DRILL_SIZE = 0.02  
    AOI_PAD_SIZE = 0.02  
    NFP_SIZE = 0.05  
}
```



```
sig_sig_pg_pg_no_shift {  
    FIRST_ROW_START_TEST_POINT = SIG  
    FIRST_ROW_NEXT_TEST_POINT = SIG  
    SECOND_ROW_START_TEST_POINT = PG  
    SECOND_ROW_NEXT_TEST_POINT = PG  
    SHIFT_ROWS = no  
    ROW_PAD2PAD_DX = 0.4  
    MAX_ROW_NUM = 2  
    MAX_COL_NUM = 6  
    PITCH = 0.3  
    TEST_LYR_SIG_APR = r100  
    TEST_LYR_PG_APR = s100  
    REF_LYR_SIG_APR = r120  
    REF_LYR_PG_APR = thr150x100x0x4x20  
    SM_LYR_SIG_APR = r115  
    SM_LYR_PG_APR = s115  
    DRILL_SIZE = 0.015  
    AOI_PAD_SIZE = 0.015  
    NFP_SIZE = 0.05  
}
```

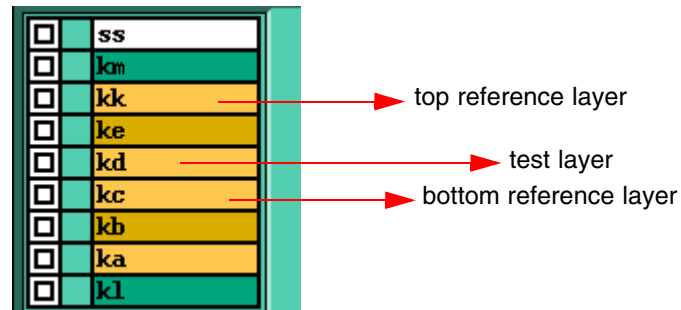
```
pg_pg_sig_sig_no_shift {  
    FIRST_ROW_START_TEST_POINT = PG  
    FIRST_ROW_NEXT_TEST_POINT = PG  
    SECOND_ROW_START_TEST_POINT = SIG  
    SECOND_ROW_NEXT_TEST_POINT = SIG  
    SHIFT_ROWS = no  
    ROW_PAD2PAD_DX = 0.4  
    MAX_ROW_NUM = 2  
    MAX_COL_NUM = 6  
    PITCH = 0.3  
    TEST_LYR_SIG_APR = r100  
    TEST_LYR_PG_APR = s100  
    REF_LYR_SIG_APR = r120  
    REF_LYR_PG_APR = thr150x100x0x4x20  
    SM_LYR_SIG_APR = r115  
    SM_LYR_PG_APR = s115  
    DRILL_SIZE = 0.015  
    AOI_PAD_SIZE = 0.015  
    NFP_SIZE = 0.05  
}
```

TUTORIAL: How to Draw a Constraint into an Impedance Coupon

TEST_LYR_SIG_APR = r100	SM_LYR_SIG_APR = r100
TEST_LYR_SIG_APR = r100	SM_LYR_PG_APR = s100
TEST_LYR_PG_APR = s100	DRILL_SIZE = 0.02
REF_LYR_SIG_APR = r100	AOI_PAD_SIZE = 0.02
REF_LYR_PG_APR = thr150x100x0x4x20	NFP_SIZE = 0.05

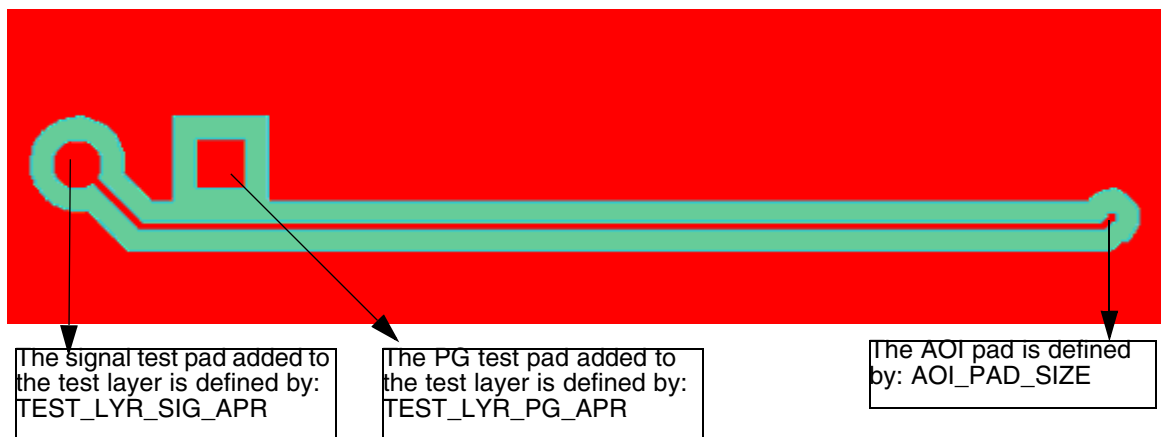
All the parameters given above define how to draw a constraint into the impedance coupon. A practical example follows.

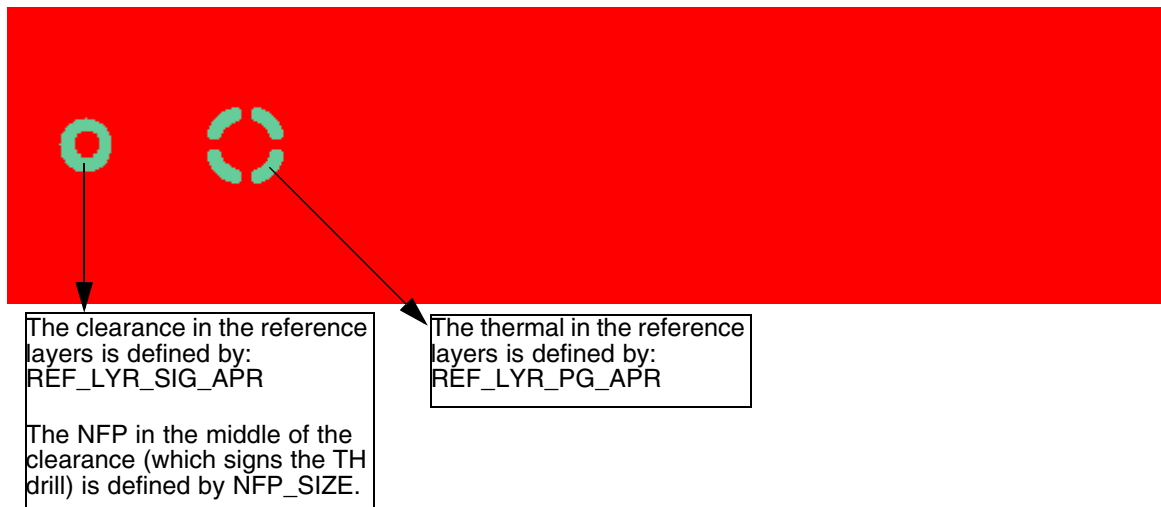
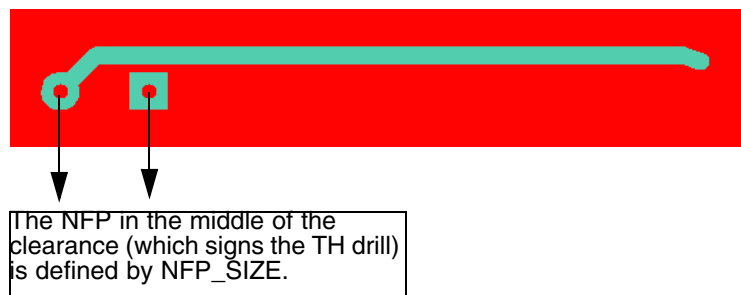
Step 1. Assume a matrix with six (6) layers, and one impedance constraint.



The constraint will be drawn to the impedance coupon as follows:

Step 2. In test layer kd, an impedance line will be added.

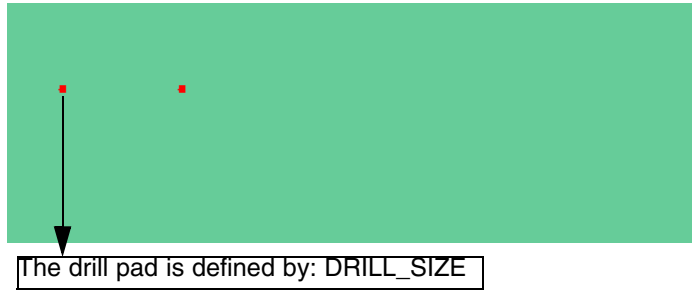
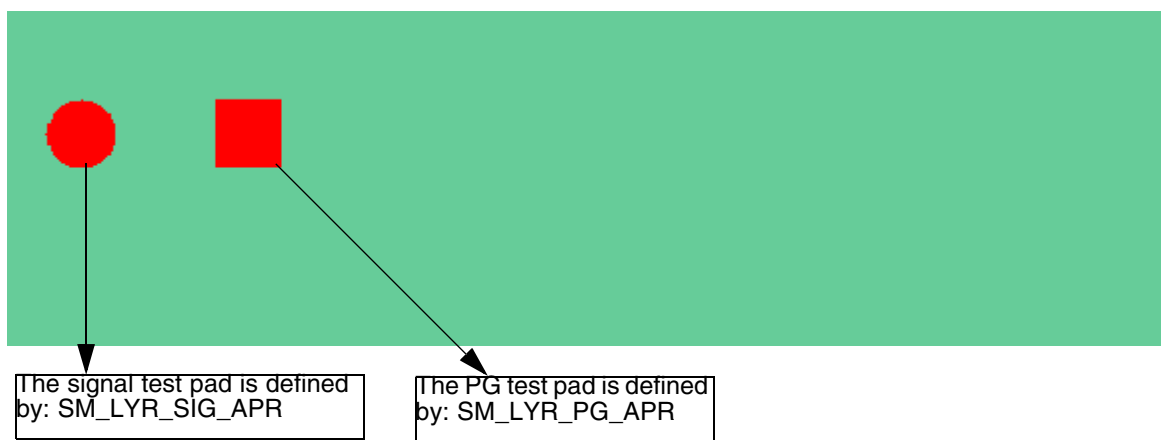


Step 3. In reference layers kk, kc:**Step 4. In the involved layers ke:**

Note The large clearance clears the area around the impedance line.

Step 5. In the uninvolved layers (like ka):

A large clearance clears the area around the impedance line (similar to layer kd). BUT this clearance is added only if it won't interfere with other constraints drawing.

Step 6. In the coupon drill layer:**Step 7. In the solder mask layers:**

The **involved** layers: (ke). (ke is a negative layer)

There is a clearance around the impedance line of layer kd. Its purpose is to imitate impedance in the PCB. The PCB impedance line on kd is covered by copper only in the ka and kk layers.



The **reference** layers (kk, ka). These layers are positive layers.



The round clearance is placed above the SIG test pad, with an NFP pad in the middle. It is connected to the SIG test pad by a drill.

The thermal is placed above the PG test pad, and it is connected to PG test pad by a drill. This drill is isolated in all layers but the reference layers.

The **uninvolved** layers (kb, ka). These are positive layers.



There is a large clearance around the impedance line in layer kd. There is no functional role for that clearance. It is only used for marking areas in which an impedance line was routed.

Step 4. The second impedance constraint is transferred to the coupon.

Test layer (ka)



Reference layer (kb). kb is a negative layer.



Uninvolved layers



Line Mode Commands

See [“New Line Mode Commands for the Impedance Coupon Generator”](#) on page 49.

Chapter 5 *Parameters, Attributes, and Commands*

New Configuration Parameters

edt_auto_f_inf_window

Type	Boolean
Default	yes
Description	<p>Determines how to view values defined for the system attribute .imp_info used in the Impedance Coupon Generator.</p> <p>Yes= By highlighting the required test pad, the information is visible in the Feature Information Popup of the Graphic Editor.</p> <p>No = A more complicated system is required.</p> <ol style="list-style-type: none"> 1. Highlight the required test pad. 2. Press the Shift key and hold it down. 3. Position the mouse on the information bar (bottom part of the editor). 4. Click M1 and release the Shift key. The Feature Information Popup opens. Here you can see all feature attributes and their values.
See Also	Graphic Editor (Doc. 0601)

iol_274x_ip_background

Type	Boolean
Default	No
Description	<p>(Released in version 9.1c)</p> <p>Determines how to interpret the IPNEG parameter which is encountered inside a RS274X file. Some CAD systems refer to it in a different manner.</p> <p>No (default) - do not add positive background to the layer in case an IPNEG parameter is encountered.</p> <p>Yes - add positive background.</p>
See Also	Graphic Editor (Doc. 0601)

New Line Mode Commands for the Impedance Coupon Generator

impcpn_open

Command	impcpn_open
Group	Impedance Coupon Generator
Description	Opens the Impedance coupon generator application, and checks out the license.

impcpn_close

Command	impcpn_close
Group	Impedance Coupon Generator
Description	Closes the Impedance coupon generator application, and releases the license.

impcpn_read_cfg

Command	impcpn_read_cfg
Group	Impedance Coupon Generator
Description	Reads the configuration file of the Impedance Coupon Generator.

impcpn_edit_imptbl_start

Command	impcpn_edit_imptbl_start
Group	Impedance Coupon Generator
Description	Clears the table of constraints of the wizard. Clears the table of constraints of the step.

impcpn_add_cnst

Command	impcpn_add_cnst
Group	Impedance Coupon Generator
Description	Reads the configuration file of the Impedance Coupon Generator
Parameter	Description
test_lyr	Test layer name
test_model	Test model name
top_ref	Top reference layer name
bot_ref	Bottom reference layer name
lw_art	Artwork line widths
lw_orig	Required line width
spc_art	Artwork spacing
spc_orig	Required spacing
impedance	Required impedance
tol	Impedance tolerance
grpId	Group number

Note All the above are constraint parameters. For more information, see [“Terminology” on page 13](#).

impcpn_edit_imptbl_end

Command	impcpn_edit_imptbl_end
Group	Impedance Coupon Generator
Description	Copies table of constraints built by the wizard to the table of constraints of the step.

impcpn_add_grp

Command	impcpn_add_grp
Group	Impedance Coupon Generator
Description	Defines a new group of constraints.
Parameter	Description
coupon	Coupon name
layout	Layout name
grp_id	Group number
ol_to_label	List of outer layers that need to be labeled. Names should be separated by ",".
ol_format	See “Text markers for pads” on page 22.
il_to_label	List of outer layers that need to be labeled. Names should be separated by ",".
il_format	See “Text markers for pads” on page 22.

impcpn_change_grp

Command	impcpn_change_grp
Group	Impedance Coupon Generator
Description	Changes the parameters of an existing group of constraints.
Parameter	Description
coupon	Coupon name
layout	Layout name
grp_id	Group number
ol_to_label	List of outer layers that need to be labeled. Names should be separated by ",".
ol_format	See “Text markers for pads” on page 22.
il_to_label	List of outer layers that need to be labeled. Names should be separated by ",".
il_format	See “Text markers for pads” on page 22.

impcpn_delete_grp

Command	impcpn_delete_grp
Group	Impedance Coupon Generator
Description	Deletes an existing group of constraints.
Parameter	Description
grpId	Group number

impcpn_set_cpn_prms

Command	impcpn_set_cpn_prms
Group	Impedance Coupon Generator
Description	Define Impedance Coupon Generator parameters
Parameter	Description
drill	Name of a drill layer
drill_size	Drill pad size. If not defined, drill size is taken from layout configuration.
add_tooling_holes	Add_tooling_holes - yes/no. If yes ICG adds tooling holes to the coupon.
plane_shield	plane_shield - yes/no. If yes, coupon shielding will be a plane shielding.
dotted_shield	dotted_shield - yes/no. If yes, coupon shielding will be a dotted shielding.
pad_sym	Symbol for dotted shielding.
pad_dx, pad_dy	dx, dy for dotted shielding.
prefix	Prefix for coupon step name
suffix	Suffix for coupon step name
split_grp	split-grp - yes/no. If yes, ICG will create as many coupons as necessary in order to allocate all required impedance constraints. If no, if ICG fails to allocate all required constraints in one coupon, it will not create additional coupons, and will return a message.
shrink	Values: none, lib, whole, half, quarter. None - Final coupon size will be the size the user defined. Lib - Coupon size may be shrunk to one of the available coupon sizes. Available coupon sizes are either the sizes defined in shrink_dims parameter, or if shrink_dims is not defined, one of the coupon sizes defined in the configuration file. Whole - Minimum coupon size is calculated, and coupon size will be shrunk to minimum, rounded to a whole number. Half - Minimum coupon size is calculated, and coupon size will be shrunk to minimum, rounded to a multiple of half. Quarter - Minimum coupon size is calculated, and coupon size will be shrunk to minimum, rounded to a multiple of quarter.

shrink_dims	List of available coupon sizes (for shrink = lib), separated by ";".
min_line	Minimal line widths (according to manufacturing limits).
min_space	Minimal space widths (according to manufacturing limits).

impcpn_generate_cpns

Command	impcpn_generate_cpns
Group	Impedance Coupon Generator
Description	Create impedance coupon according to constraints and parameters defined by previous line mode commands.

New System Attributes

.imp_info

The attribute describes the impedance constraints for the source step. For more information, see [“On-Screen display” on page 27](#).

.detch_smooth

Smoothing lines are marked with the new system attribute **.detch_smooth**

New Line Mode Commands to Control Access to Scripts

The following new line mode commands apply to running/hiding the script generator.

script_debug_hide

Command	script_debug_hide
Group	Graphic Editor
Description	Hides the C-shell debugger

script_debug_show

Command	script_debug_show
Group	Graphic Editor
Description	Shows the C-shell debugger

script_bind_show

Command	script_bind_show
Group	Graphic Editor
Description	Shows the script binding window

script_bind_hide

Command	script_bind_hide
Group	Graphic Editor
Description	Hides the script binding window

script_run_show

Command	script_run_show
Group	Graphic Editor
Description	Shows the script run window

script_run_hide

Command	script_run_hide
Group	Graphic Editor
Description	Hides the script run window

*New Line Mode Commands for Orbotech AOI Interface**cdr_get_flow_stage_status*

Command	cdr_get_flow_stage_status
Group	Orbotech AOI Manager
Description	The command is used for finding the red/green status of a flow stage.
Returns	green or red
Parameter	Value
flow_stage	Stage of flow. Values: prms,stage,frs,inspect, exclude, align_tgts, alignment, vrs_tgts

cdr_get_step

Command	cdr_get_step
Group	Orbotech AOI Manager
Description	The command is used for getting the name of the step loaded in the cdr interface
Parameter	Value
step name	

cdr_get_machine

Command	cdr_get_machine
Group	Orbotech AOI Manager
Description	The command is used for getting the name of the machine loaded in the cdr interface
Returns	machine name

Parameter	Value

cdr_get_nom_line

Command	cdr_get_nom_line
Group	Orbotech AOI Manager
Description	The command activates an analysis that calculates the layer's nominal line and returns it. Note: Requires auto setup license
Returns	Line width in mils or microns - depending on AOI interface units
Parameter	Value
layer	name of an existing layer

cdr_get_nom_space

Command	cdr_get_nom_space
Group	Orbotech AOI Manager
Description	The command activated an analysis that calculates the layer's nominal spacing and returns it.
Returns	Nominal spacing of the type required in mils or microns - depending on AOI interface units. Note: Requires auto setup license
Parameter	Value
layer	name of an existing layer
space type	Options: nom_space; nom_nfp_space; nom_plane_space.

Chapter 6 *KIT List*

42 KIT items

KIT No.	Description
12036	Wrong colors in the Graphic Snapshot / Ctrl+P.
13001	How to disable "Script" "Script Debugger" menu item for unnecessary user
13084	Break Arcs Cause SIP problem
13099	Set executing priority on DEC
14301	Manual input does not work for excellon files
14361	Set SMD attribute DFM - problem in MVia mode
14443	Add "smoothen lines" attribute in DETCH.
14535	Front test point/s missing in Hioki netlist information file
14589	TearDrop phenomena/problem
14613	Galvanic Etch: Enlarge the gold pads
14627	Problem in DMF Tangency Elimination
14656	Orthogonal stretch of negative lines
14684	Select board net problem
14692	Pannel_Pattern_error
14718	Copy misc layer problem in flip step
14749	"Add teardrop" deletes a trace
14753	Illegal plated drills
14769	Output ESI laser format would lose some arc.
14770	Input DXF file would lose pad.
14771	Copper removed automatically
14778	Genesis Job Delete and Save Issues
14780	Some pads have gone missing using 274x output
14784	Not possible to remove midpoints anymore
14790	Some lines are not compensated for no reason
14792	Input DPF jobs take too long.
14794	Microcraft Output and .et_align Attribute
14796	Hitachi S&R input
14797	Failure of dxf input
14801	Crash when Filtering by Net Name on G.E.
14806	Warning on startup/opening a job
14808	Navigation and display drill tool is not working
14814	Input zero degree arc problem

KIT No.	Description
14816	DI OUTPUT crash
14817	Unmerge cause attributes loss
14818	Input zero degree arc problem
14826	Input hpg format problem
14833	DTM problem
14840	Ver_91c1 Panelization
14844	Question about split by DEC
14850	Internal Error-Assertion failed: npt == npt_top
14865	Problem with IPC output: same point belongs to 2 different nets
14872	Illegally quit when using ADM
