



Design Rules

© Norlinvest Ltd, BVI. Visionics is a trade name of Norlinvest Ltd. All Rights Reserved. No part of the Design Rules document can be reproduced in any form or by any means without the prior written permission of Visionics. Design Rules document is subjected to change without notice. Visionics will make changes in a manner that will not affect dependent systems.

Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Visionics.

Visionics, EDWinXP, Docone, EDCOMX, SimWinXP and Mixed Mode Simulator and their respective logos are trademarks or registered trademarks of Visionics. Unauthorized duplication of this work may also be prohibited by local statute.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Visionics. The information contained herein is the proprietary and confidential information of Visionics or its licensors, and is supplied subject to, and may be used only by Visionics's customer in accordance with, a written agreement between Visionics and its customer. Except as may be explicitly set forth in such agreement, Visionics does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Visionics does not warrant that use of such information will not infringe any third party rights, nor does Visionics assume any liability for damages or costs of any kind that may result from use of such information.

CONTENTS

CONTENTS	2
DESIGN RULES	3
OPERATION.....	3
ROUTING LAYERS.....	3
VIA RULES	5
TRACE RULES	7
CLEARANCES	8
AUTOROUTER	8
LOCKED EDIT	9
LAYOUT COMPONENT PLACEMENT.....	9

Design Rules


Pops up EDWinXP – DRC Setup to set certain default Design Rules to work with EDWinXP. This utility is used to preset certain parameters for manual, semiautomatic and automatic routing of traces and component placement. With the given parameters, the Layout conducts a design rule check and any problems are marked by an Error label. The errors may be queried using the Redraw/ Error. The design rule settings may be saved as a permanent setting or may be set for the circuit or for the current project only. The design rule settings may be saved as a permanent setting or may be set for the circuit or for the current project only.

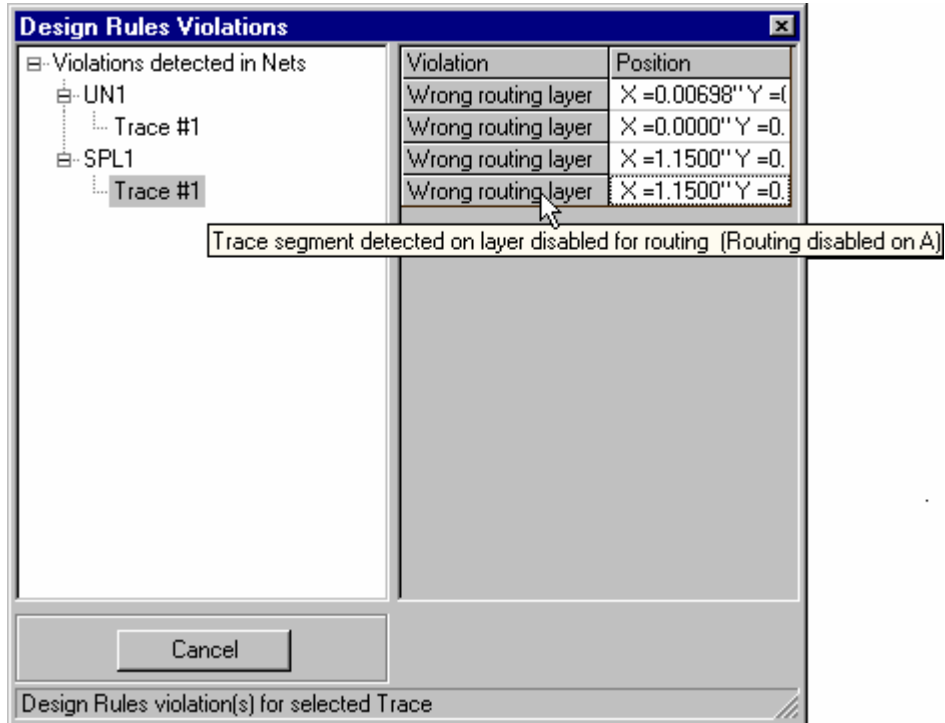
Operation

Select **EDWinXP → Project Explorer → Design Rules** and set the required settings. The details regarding the parameters that may be set are given below.

1. **Routing Layers** (Applies to Layout Editor Only):
2. **Via Rules** (Applies to Layout Editor Only):
3. **Trace Rules** (Applies to Layout Editor Only):
4. **Clearances** (Applies to Layout Editor Only):
5. **Autorouter** (Applies to Autorouter Only):
6. **Locked Edit** (Applies to Layout and Schematic Editors Only)
7. **Layout component Placement** (Applies to Layout Editor Only)

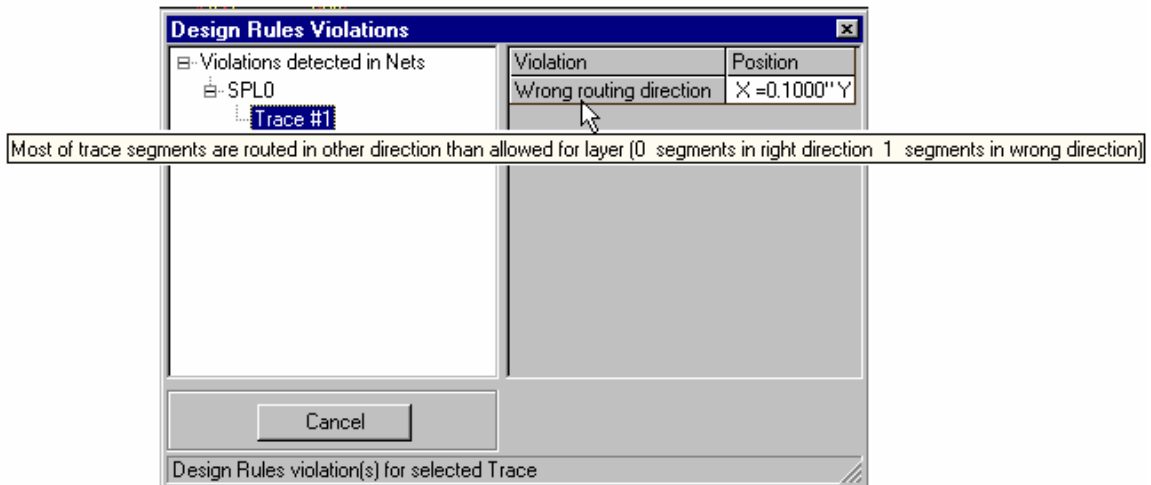
Routing Layers

Not Used:-If this option is selected the designer is not allowed to perform routing in this layer. A message box pops up prompting to switch to the allowed layer. Nevertheless, user may use the option tool  **Place On Selected Layer** and switch to the layer which the user has set to **Not Used**. After setting this option to NOT USED, user may validate the design for any design violation. This is done using the Layout Editor →Auto/ Autocheck→Check Other Violation tab. In this tab check **Traces on Wrong layers**. Immediately the results are displayed in the Design Rule Violations dialog. Click on the right side row to display the description of the error. When clicked on each error, the error portion is highlighted on the workspace.



Horizontal or Vertical:

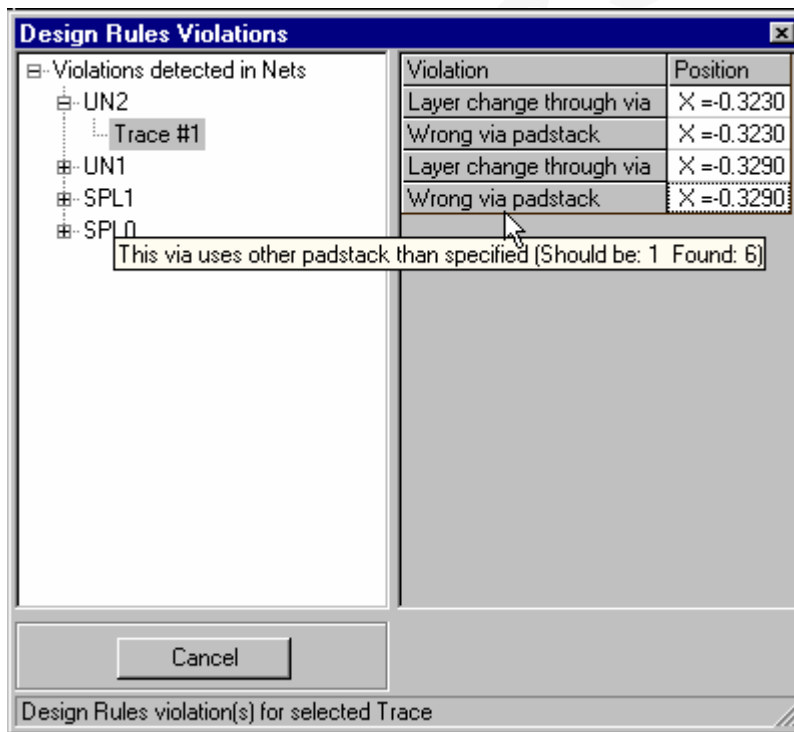
Set the required direction and check the option **Traces routed on wrong direction** in Autocheck window (Layout Editor → Auto/ Autocheck → Check Other Violation tab). Execute the operation to display the results in the Design Rule Violations dialog window. In this window, click on the right side row to display the description of the error. When clicked on each error, the corresponding error is highlighted on the workspace.



Via Rules

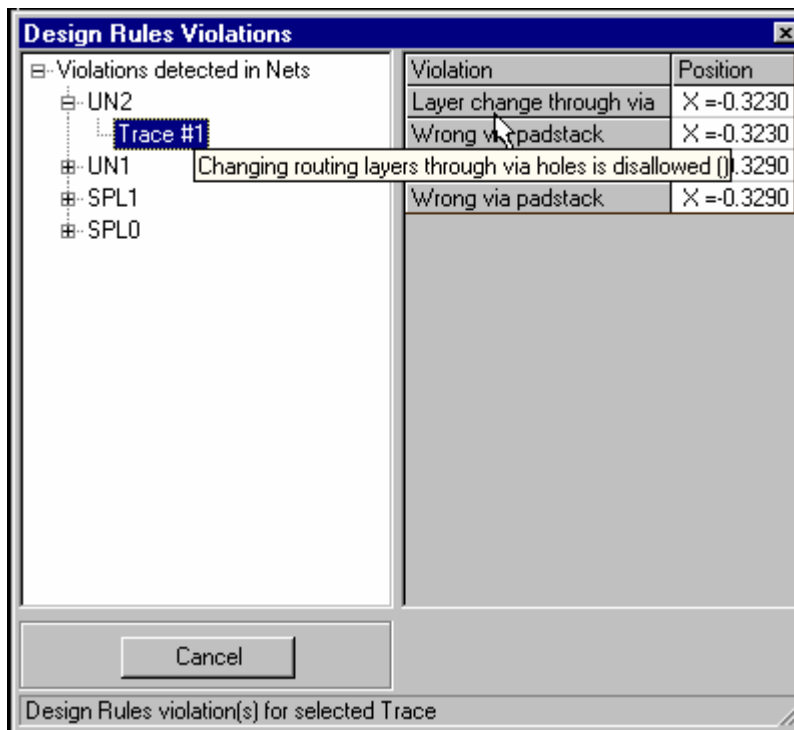
Via Padstack number:

Allows to set the required Via Padstack. If any other via padstack is used, system displays the violation error when Autocheck (Layout Editor →Auto/ Autocheck→Check Other Violation tab) is performed provided the option **Wrong Via Hole Padstack** is checked. The errors are displayed in the DR Violation dialog as shown below.



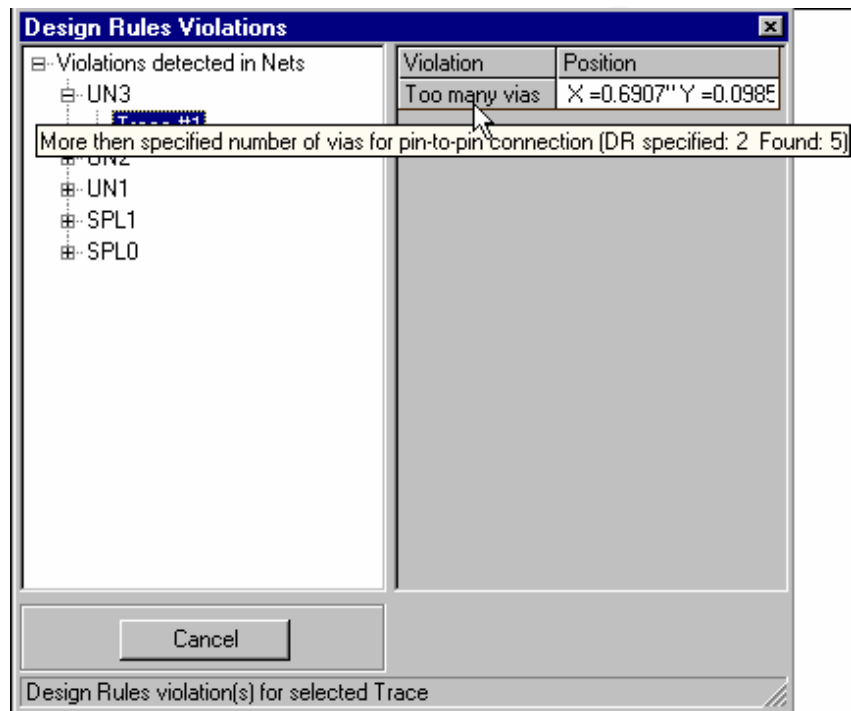
Via allowed:

Allows to specify whether via padstack can be used in the design or not. If not specified, system displays the violation error when Autocheck (Layout Editor →Auto/ Autocheck→Check Other Violation tab) is performed provided the option **Layer Change Through Via Hole** is checked. The errors are displayed in the DR Violation dialog as shown below.



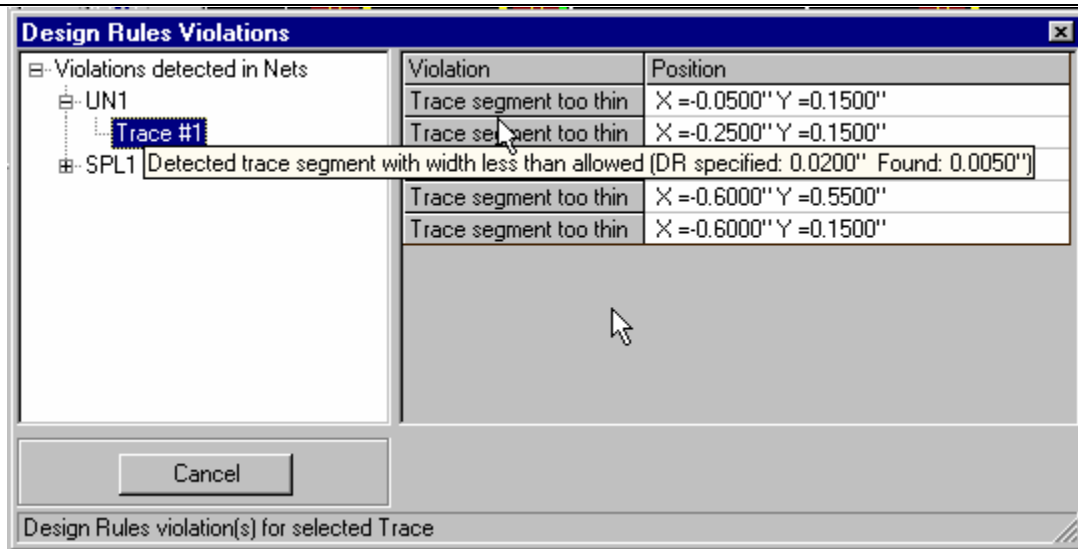
Maximum number of Vias per Trace

Permits the user to specify the maximum number of vias allowed on a trace connecting between component Pads. If the number of via padstack used is more than the number specified in DRC dialog, system displays the violation error when Autocheck (Layout Editor → Auto/ Autocheck → Check Other Violation tab) is performed provided the option against **Too many Vias on Pin to Pin Connections** is checked. The errors are displayed in the DR Violation dialog as shown below.



Trace Rules

This option allows to set the required trace parameters such as trace width, trace airgap, and the trace length. The value of certain trace parameters may be selected from the given list or may be specified. If any of these settings is violated, system displays the violation error when Autocheck (Layout Editor | Auto | Autocheck | Check Other Violation tab) is performed provided the required option under **Trace Rules** is checked. The errors are displayed in the DR Violation dialog as shown below.



Clearances

This option allows to set the clearances between Trace to Trace, Pad to Pad, Pad to Trace and Single Trace Check Width. To check any violation in the design, run the Autocheck (Layout Editor | Auto | Autocheck | Check Clearances tab). To use the Design Rules settings, check **Use Clearances as in Design Rules for Layers**. The error labels are displayed on the design. Each error labels signify the type of error encountered.

Autorouter

The settings for the Arizona Autorouter may be set in DRC dialog. The settings include Fanouts routing enabled, Trace routing enabled, SMD connection only, Number of routing axis, Wrong layer direction cost, Wrong direction optimizing cost, Right direction optimizing cost and Vias Optimizing cost. These settings may also be set in Arizona Autorouter however, to use DRC settings check the necessary options in the Arizona autorouter. The options available in Arizona are

- ☉ **Use these settings if not explicitly specified in Design Rules:** Allows to use the settings specified in Arizona if the settings are not explicitly specified in DESIGN RULES.
- ☉ **Override setting specified in Design Rules:** Allows to override the settings set in DESIGN RULES.

Locked Edit

This option is applicable to both Layout and Schematic editor. Certain operations related to Nets, Nodes, Pin Swapping, Adding/ Deleting components may be locked by enabling the check boxes. The options set prompts the user of rules violations thus prohibiting the user to violate the rule.

Layout component Placement

The options listed here are explicit to **layout components**. The parameters include component orientation, name placement, name orientation, component name distance, component snap, distance to surrounding layout components and check distance while placing. The options set prompts the user of rules violations thus prohibiting the user to violate the rule. The Layout Component Snap works a slight differently. Follow the description given below.

Component Snap (applicable to layout only)

In earlier version, snap value may be enabled or disabled by clicking on the icon of snap. However, in this release snap has been improved to suite best the environment. In addition to the snap value set in the layout module, it may also be set in Design Rule Check dialog also. In this case, the snap value set in Default Design Rule dialog has higher priority over the value set in layout editor. A typical example follows.

Consider a situation where two snap values are required. This kind of situation occurs when certain components require to be relocated with snap value say .1000" whereas certain other components require snap value say .0500". The snap value that is required for most of the components can be set in Design Rule Check dialog. By default, the user may now work with the value set in Design rule. To use the snap value set in Layout Editor, the user may use CTRL key and work with the current settings in Layout Editor.

Operation

Snap values are listed in the Snap text box. The required value may be selected from the down list box or manually entered in the text edit box provided. When unit (View | Unit) is switched between inches and millimeters, the value in Snap changes accordingly.

How to use an alternate Snap value (for Layout Components only) using design rule setup dialog:

Open Design Rule dialog from EDWinXP Project Explorer and enter the snap value say .1000" in Layout component Placement. In Layout Editor, set the snap value to .0500". Now try relocating the components, you may find that the components move with Snap value .1000". To use the Snap value selected in Layout editor, press CTRL key and try to relocate. Now the Layout Components move with Snap value .0500".