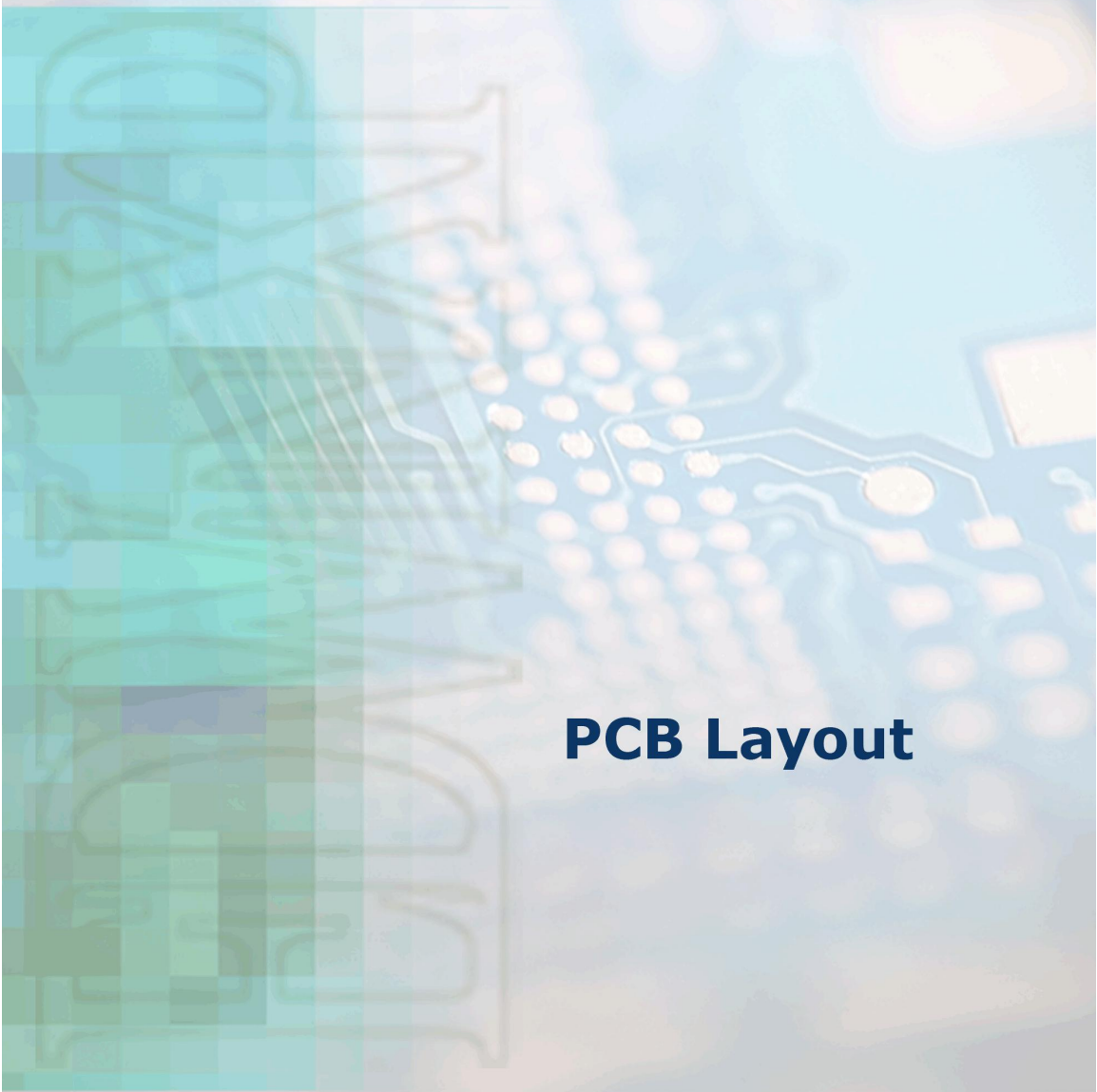


Electronic Design for Windows
EDWINXP



PCB Layout

VISIONICS

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

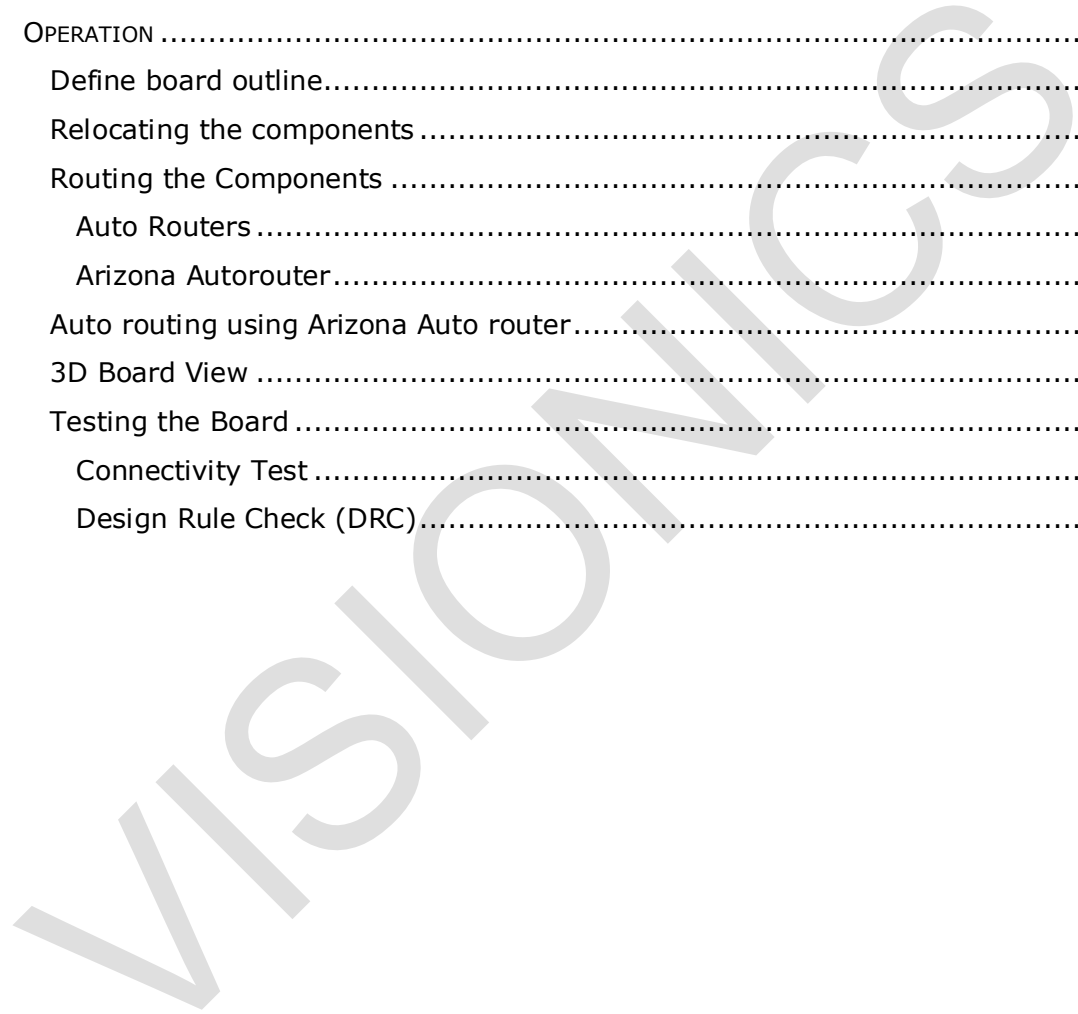
Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Visionics.

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

PCB LAYOUT	4
OPERATION	4
Define board outline.....	4
Relocating the components	5
Routing the Components	6
Auto Routers	7
Arizona Autorouter.....	7
Auto routing using Arizona Auto router.....	7
3D Board View	9
Testing the Board	9
Connectivity Test	9
Design Rule Check (DRC).....	10



PCB LAYOUT

Layout Editor is used to design the PCB layout of a circuit. The project database supports design of 32-layers boards (28 trace layers, 2 silk-screen & 2 solder masks). The design can be captured either in schematic capture or directly in layout editor. In the former case, the design is front annotated to the layout editor and in the latter case it is back annotated to schematic editor.

Components are automatically created as a result of packaging executed while editing the schematic diagram of the circuit. Layout may be started without a preexisting schematic. In that case, new parts may be also inserted on the board in similar fashion as in Schematic Editor. Location and orientation of components is defined either by manual relocation to desired position or with help of auto placer. Traces may be laid manually with automatic via insertion whenever routing layer is changed. Six types of user defined via pads are supported. Editor includes also a number of online automatic functions to route/ reroute traces for single nets and to reroute existing traces for relocated components. Dedicated full board auto router module is integrated with PCB Layout Editor. Copper pour areas are defined as polygons and may be placed on any trace layer. Insertion of air gaps and thermal pads on artworks is automatic.

Design rules for manual, semi-automatic and automatic routing of traces and component placement are user defined and may be set individually for each project. Design rules violation, clearance errors and missing or incomplete connections are detected automatically.

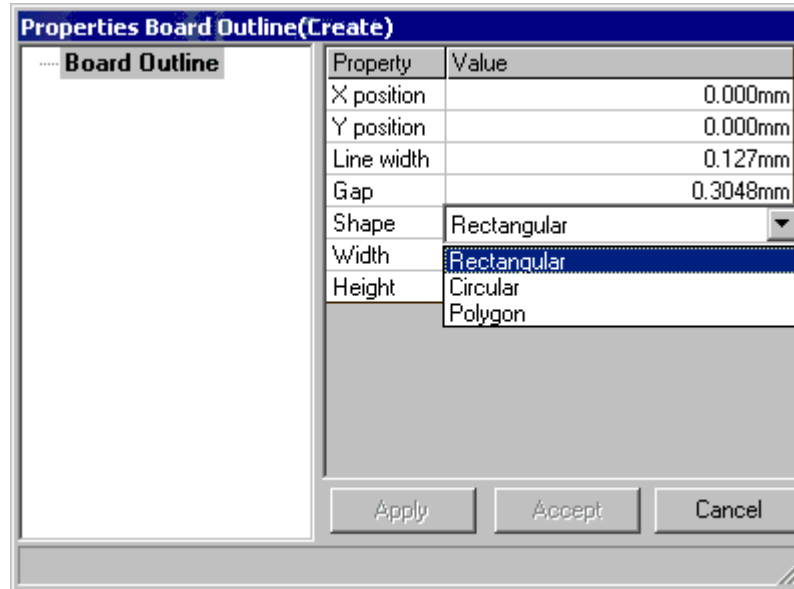
Operation

1. **PCB Layout** is invoked from Project Explorer in the following ways.
2. Right click **PCB Layout** and select **Edit PCB Layout** from the list. Or Select **Edit PCB Layout** from the task list or from the task toolbar.

Define board outline

1. Select **Tools** → **Board Outline**

2. Select **Define Outline** (function tool) → **Create Board** (option tool). This tool enables free hand drawing, using which a board of desired shape can be drawn on the workspace.
3. To select a pre-defined board format, select the option tool **Textual Mode**. The **Properties Board Outline (Create)** dialog pops up.
4. Enter the desired values and click **Accept** button.



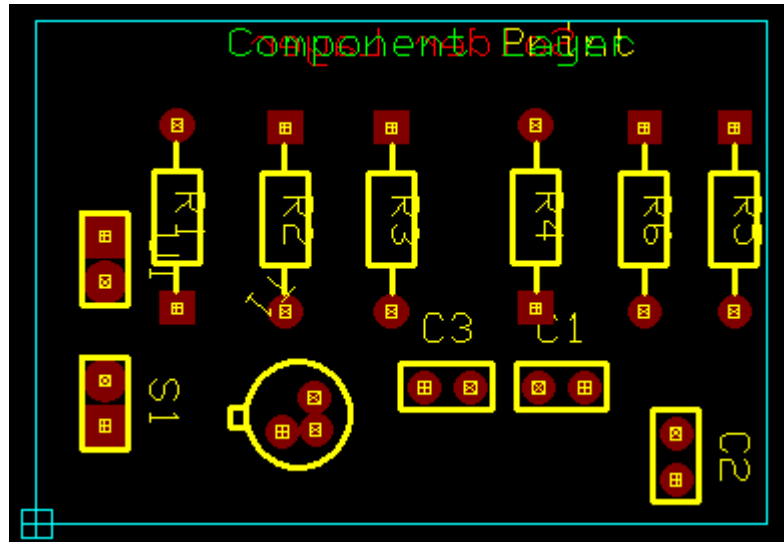
Relocating the components

The components placed on the schematic, which contain both symbol and package, are front annotated to the layout after packaging. These components are positioned at board datum (0,0) automatically and may be relocated either manually or automatically

Note: *Only parts that contain both symbol and package will be automatically back annotated to the schematic.*

1. Before we start loading Parts on to the page, turn ON **Grid** by enabling grid from the dropdown, in Standard Toolbar. The value for grid may be selected from the drop down list as .1000".
2. Similarly, set **Snap** value to .0500" for better placement of the components.
3. Select **Tools** → **Components** → **Relocate Component** (function tool) to relocate the components.

4. Enable **Ratsnest** (**F7** key) option tool of **Relocate Component** function tool to view ratsnest while relocating the components to ensure that components having a large number of interconnections are positioned close to each other. Pressing **shift** key while relocating/ stretching an item allows the item to move/ stretch smoothly.






Routing the Components

Connections between components may be established by using **Traces** and **Copper Pour** areas.

1. Select **Tools** → **Connections** → **Route** (Function tool) to enable a set of Routing tools on right clicking the workspace.
2. Adjust **zoom precision** to view the pins properly.
3. Turn on **Preference/ Guidelines (Next unconnected node)** before routing because this option guides you to take the easiest path to route.
4. Select **True Size** and **Pad frames** from **View/ Layout**, enabling you to select proper trace size. This also prevents you from creating errors such as traces crossing over pad, traces very close together etc.
5. First route **power** and **ground signals** (Net SPL0 and +5V). In Project Explorer, select Project / Project Design Rules and set the routing width to **0.030"** for Pwr/Gnd lines and **0.013"** for Signal lines.
6. To start routing traces, first select the power points

7. Select layer for routing from **Layers** in the main menu. 28 layers are available for routing. By default **COMP LAYER** is selected. Click on pin to route on **SOLD LAYER**.

 **Tips:** While routing, enable the tool  **Snap Trace by 45 degree** to change routing directions in steps of 45 deg only.

8. Move cursor with 45° angle through a short distance and click at the nearest point.
9. Terminate routing of the trace by pressing END or F4 key on your keyboard. Or click on the tool  **End Connections**.

Auto Routers

The three Auto routers available are:

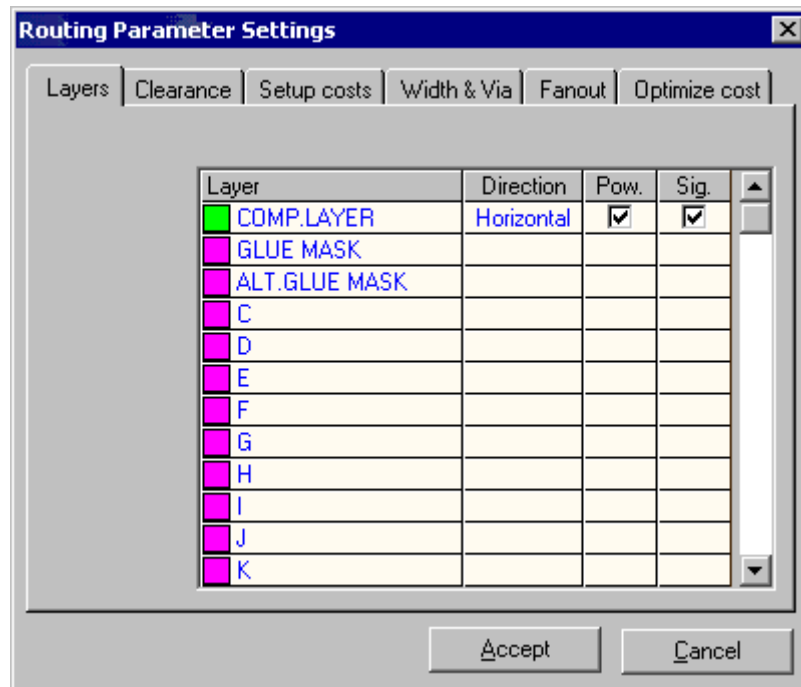
1. Autorouter-Arizona
2. Autorouter-Spectra
3. Autorouter-Maxroute

Arizona Autorouter

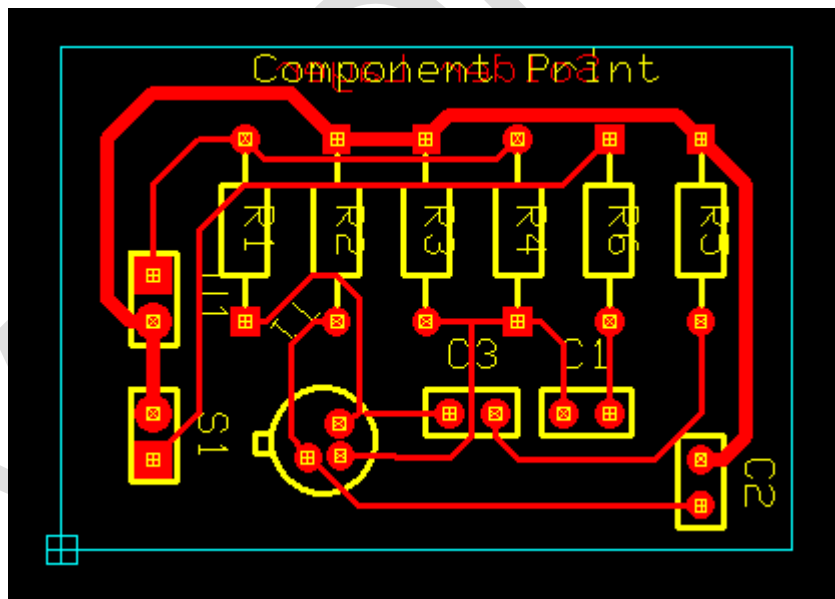
Arizona auto router is an integrated module of the EDWinXP. It uses its own temporary project and simplified graphics. The Arizona auto router allows routing the traces of a PCB Layout automatically.

Auto routing using Arizona Auto router

1. Select **Auto → Auto Router → Arizona**.
2. Select **File → Load board to route from project**.
3. Select **Parameters Setup** (function tool) → **Routing Parameter Settings** (option tool).
4. Check **Solder Layer** in the **Routing Parameters** Settings window



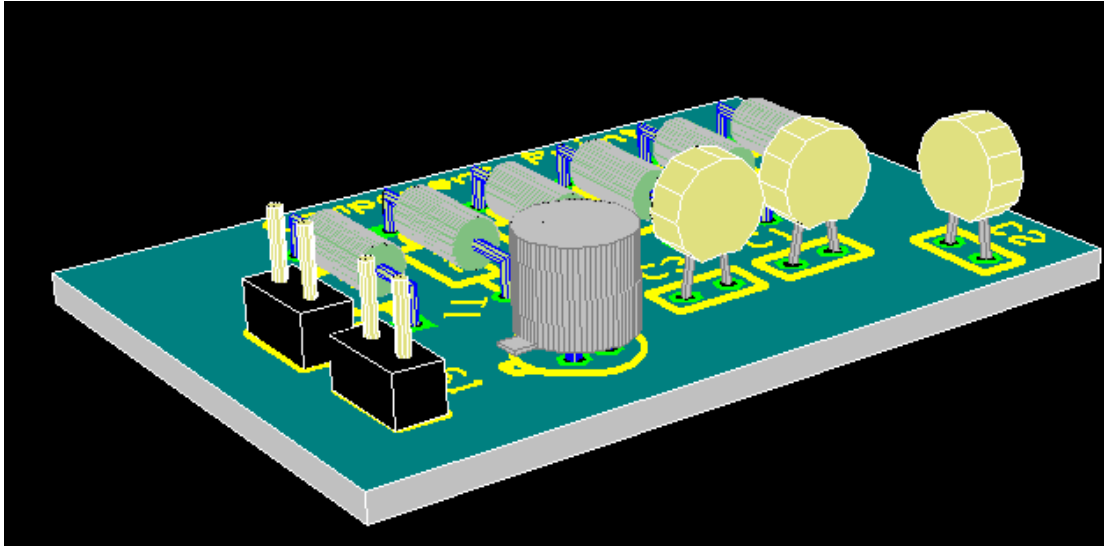
5. Select **Auto routing Routines** (function tool) → **Start Auto Router**.
6. Select **Auto routing Routines** → **Miter**.



7. Close **Arizona Auto router** window, Click **Update project and Exit** to save the project.

3D Board View

To view the 3D board view in the Layout Editor select **Tools → 3D Board Viewer**



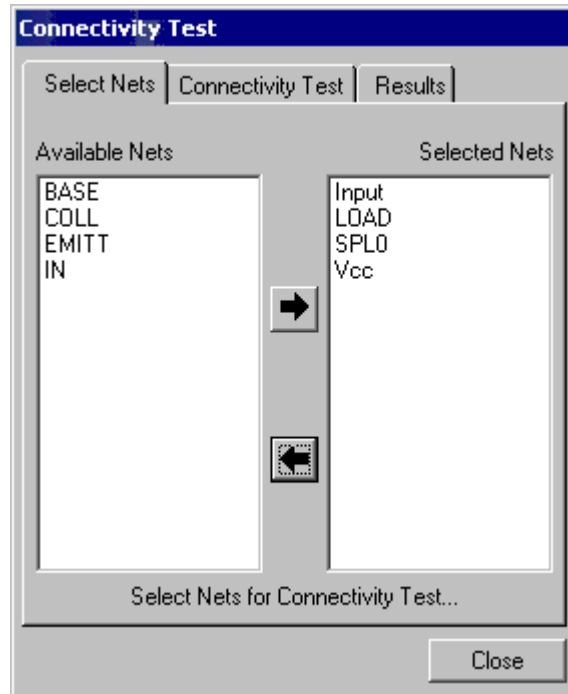
Testing the Board

While designing a PCB, it is quite obvious that a number of errors may occur. These errors may be in the form of overlapping pads, unconnected Nodes, traces crossing another trace, etc. Such errors must be taken care of before printing the PCB. To find out such errors, certain checks on the board are done. **Connectivity and DRC check** are the two checks.

Connectivity Test

Connectivity test may be used to check whether there is any electrical discontinuity (unconnected nodes or deleted trace segments) in a single net. By setting certain parameters, the test can be performed either on individual nets or all the selected nets.

1. Select **Tools → Connections → Connection Property → Test Connectivity**



2. Click on the board or on a net. The **Connectivity Test** dialog pops up.
3. Select the required nets using the move keys and click the connectivity test tab.
4. Select **Connectivity test** tab, set anyone of the three options:
 - Stop at first fail:** Test stops at the first occurrence of error.
 - Test all selected Nets:** Checks whether all **Nodes** of the selected **Nets** are connected.
 - Test single Net:** Checks connectivity of the **Nodes** of the selected **Net**.
5. Click the **Test** button to display the results.
6. Perform this test until "**Tested Nets – Fully Connected**" message is displayed in this window.

Design Rule Check (DRC)

This utility is used to create an error free board to enhance the efficiency of your board. It automatically smoothes, miters, and checks for both aesthetic and manufacturing problems that might have been created in the process of manual or automatic routing. This test helps us to check the clearance between pad to pad, pad to trace and trace to trace.

1. Select **Autocheck** from **Auto** Menu. The **Autocheck Setup** dialog pops up. Select Check Clearances tab.

2. Select the layers and enter the clearance value in the window.
3. To select all the layers used in the project click the **Set To Used** button.
4. Click **Execute**.

