

Electronic Design for Windows  
**EDWINXP**



# Signal Integrity Analyzer

***VISIONICS***

© Norlinvest Ltd, BVI. Visionics is a trade name of Norlinvest Ltd. All Rights Reserved.

No part of the Signal Integrity Analyzer document can be reproduced in any form or by any means without the prior written permission of Visionics. Signal Integrity Analyzer 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.



## Signal Integrity Analyzer

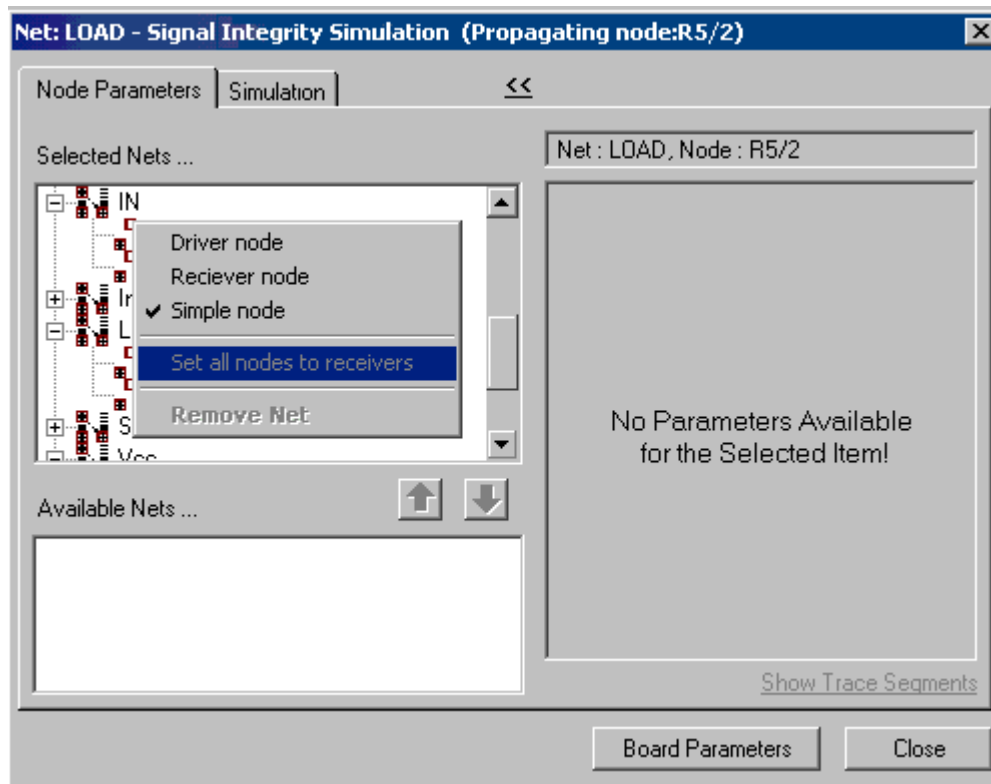
The Signal Integrity Analyzer examines the probable distortion of high-speed signals as they pass through traces on the PCB. The results of the analysis are displayed in the Waveform Viewer. The purpose is to predict how a signal deviates from its ideal (or intended) behavior in a real-world setting.

### Steps for Signal Integrity Analysis

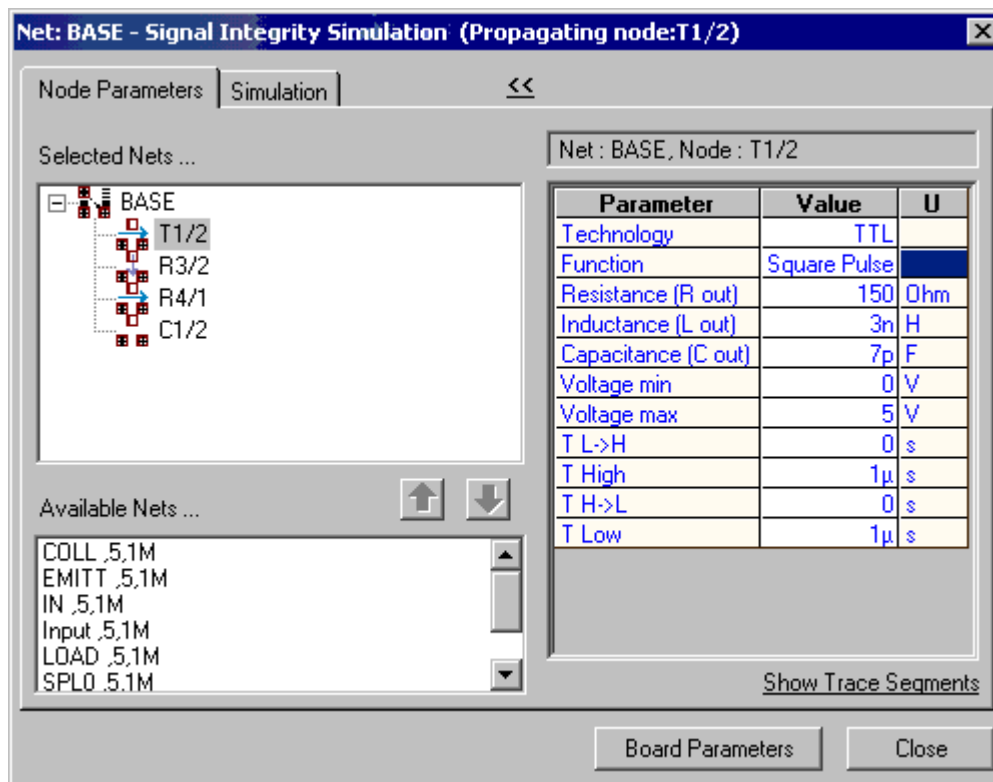
Under normal working conditions, signals propagate from some pins of the components in the circuit (technically called **Driver Nodes**), through the interconnecting traces and into pins on the same or other components (called **Receiver Nodes**).

This analysis detects the degree of distortion of the signal from its ideal behavior while it passes through the trace and graphically presents the comparison result with the help of the **Waveform Viewer**.

1. Right click on the **PCB LAYOUT** in the Project Explorer, select **Board Analyzer**. Select the **Electromagnetic Analyzer** tab.
2. Right click on the workspace and select the tool  **Signal Integrity / Field Analyzer**, and click on any of the trace, which is to be analyzed.
3. Now select a net. A window pops up where you may set all the parameters for simulation. To check the integrity of an electromagnetic signal while it passes through this trace, go through the following steps.
4. All nets are displayed in the frame Available Nets. Select a net from the list and click . The segment nodes of the selected net get displayed in the frame. Right click on any of the node and set it as driver node and another node as receiver node.



5. The electrical properties of the trace will influence the signal as it passes through it and this is shown in the respective columns. The Trace Segment parameters of the driver node may be viewed by clicking on **Show Trace Segments**.



6. Set the following parameters.

**Max voltage to 5V**

**T L->H = 0**

**T H = 1us**

**T H-> L = 0**

**T L = 1us**

7. Set TTL technology to the receiver nodes. Check the check box corresponding to Test Points. The **Test points** allow displaying the output of the selected node in the waveform viewer. If no test point is selected then output waveform alone is displayed on the waveform viewer.
8. To execute simulation, Start time and End time of the simulation as well as the sampling time interval must be specified. This corresponds to the parameters Start time, Time limit and Time step. Enter 1n for Time Step and 2µs for Time limit.
9. Click on **Calculate** button. The **Calculate** button allows to calculate the maximum time limit and will update the text box Time limit. Here the time step is set to **1ns** and Time limit to **2us**.

10. Check the box **Generate Waveform**. Click the **Simulate** button to process the simulation.
11. The waveform obtained after simulation is shown below.

