

# Signal Integrity Toolbox™

Reference



# MATLAB®

R2022a



# How to Contact MathWorks



Latest news: [www.mathworks.com](http://www.mathworks.com)  
Sales and services: [www.mathworks.com/sales\\_and\\_services](http://www.mathworks.com/sales_and_services)  
User community: [www.mathworks.com/matlabcentral](http://www.mathworks.com/matlabcentral)  
Technical support: [www.mathworks.com/support/contact\\_us](http://www.mathworks.com/support/contact_us)



Phone: 508-647-7000



The MathWorks, Inc.  
1 Apple Hill Drive  
Natick, MA 01760-2098

## *Signal Integrity Toolbox™ Reference*

© COPYRIGHT 2021–2022 by The MathWorks, Inc.

The software described in this document is furnished under a license agreement. The software may be used or copied only under the terms of the license agreement. No part of this manual may be photocopied or reproduced in any form without prior written consent from The MathWorks, Inc.

FEDERAL ACQUISITION: This provision applies to all acquisitions of the Program and Documentation by, for, or through the federal government of the United States. By accepting delivery of the Program or Documentation, the government hereby agrees that this software or documentation qualifies as commercial computer software or commercial computer software documentation as such terms are used or defined in FAR 12.212, DFARS Part 227.72, and DFARS 252.227-7014. Accordingly, the terms and conditions of this Agreement and only those rights specified in this Agreement, shall pertain to and govern the use, modification, reproduction, release, performance, display, and disclosure of the Program and Documentation by the federal government (or other entity acquiring for or through the federal government) and shall supersede any conflicting contractual terms or conditions. If this License fails to meet the government's needs or is inconsistent in any respect with federal procurement law, the government agrees to return the Program and Documentation, unused, to The MathWorks, Inc.

## **Trademarks**

MATLAB and Simulink are registered trademarks of The MathWorks, Inc. See [www.mathworks.com/trademarks](http://www.mathworks.com/trademarks) for a list of additional trademarks. Other product or brand names may be trademarks or registered trademarks of their respective holders.

## **Patents**

MathWorks products are protected by one or more U.S. patents. Please see [www.mathworks.com/patents](http://www.mathworks.com/patents) for more information.

## **Revision History**

September 2021	Online only	New for Version 1.0 (Release 2021b)
March 2022	Online only	Revised for Version 1.1 (Release 2022a)

<b>1</b>	<u>Apps</u>
----------	-------------



# Apps

---

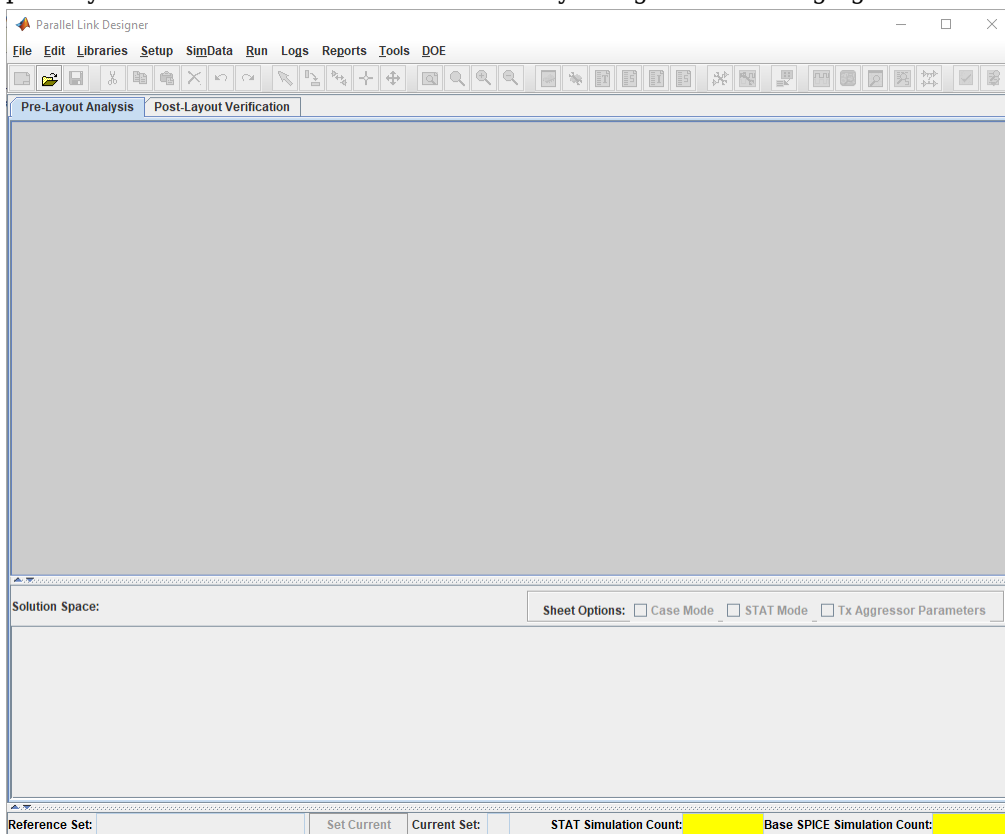
# Parallel Link Designer

Analyze PCB designs for parallel link applications

## Description

The **Parallel Link Designer** analyzes PCB designs for parallel link applications for both pre-layout and post-layout simulations. To perform post-layout analysis, you need a license for RF PCB Toolbox™.

Using pre-layout analysis, you can determine system-level noise and timing margins from integrated signal integrity, waveform, timing, and crosstalk analysis. The pre-layout analysis environment generates design guidelines for your board layouts, package layouts, connectors, and cabling. The post layout environment verifies the actual layout against the design guidelines.



## Open the Parallel Link Designer App

- MATLAB® Toolstrip: In the **Apps** tab, under **Signal Processing and Communications**, click the app icon.
- MATLAB command prompt: Enter `parallellinkdesigner`.

## Examples

- “Analyze Parallel Links with Parallel Link Designer”
- “Configure DDR Controller with Two Memory Designators”

## Programmatic Use

`parallelLinkDesigner` opens a new session of the **Parallel Link Designer** app, enabling you to design and analyze a parallel link.

`parallelLinkDesigner(path/file.edk)` opens the interface designed by the file.

`parallelLinkDesigner(file.script)` runs a script file and returns the app process handle.

## More About

### Simulation Parameters

Simulation Parameters control how the analysis is run in the **Parallel Link Designer** app. For detailed information about them, see “Simulation Parameters Used in Parallel Link Design”.

### Corner Conditions

There are two types of corner conditions you can vary using the **Parallel Link Designer** app.

- IC Environment Corners — These are the temperature parameter for each corner. They do not affect IBIS buffer models.
- Etch Corners — These are the scaling factors for the  $Z_0$  and  $T_{pd}$  parameters of transmission line models. Both ideal and lossy transmission line models are scaled. Lossy transmission line models are scaled by computing the values of  $Z_0$  and  $T_{pd}$  from the typical corner L and C values.

For more information, see “Specify Corner Conditions in Parallel Link Design”.

### Stimulus Patterns

You can specify the stimulus patterns for the time domain analysis. If the specified pattern for a designator has fewer bits than the simulation length, the pattern is repeated from the first bit of the pattern. If the pattern is longer than the simulation length the simulation will end at the time specified by **Time Domain Stop**. For more information, see “Stimulus Patterns in Parallel Link Design”.

### Widebus (Pre-Layout Crosstalk Analysis)

In **Parallel Link Designer** app, pre-layout crosstalk analysis is referred to as widebus crosstalk analysis. It consists of single transfer sheets (transfer nets) and a “widebus” or coupled sheet with a victim transfer net and one or more aggressor nets.

Each unique transfer net on a widebus schematic sheet is known as a “widebus group”. That group consists of ONLY the transfer net designators (drivers, receivers or I/O buffers). On a widebus sheet any widebus group can be a victim or aggressor and multiple instances of any widebus group can be placed on the schematic. The widebus groups inherit the designator properties defined in their

transfer net sheet. Coupled w-lines, connectors, package models or s-parameter blocks can then be used in the widebus schematic to connect up each widebus group. One of the instances of a widebus group will be designated as the victim net on the schematic.

Each unique transfer net on a widebus schematic sheet is known as a “widebus group”. That group consists of ONLY the transfer net designators (drivers, receivers or I/O buffers). On a widebus sheet any widebus group can be a victim or aggressor and multiple instances of any widebus group can be placed on the schematic. The widebus groups inherit the designator properties defined in their transfer net sheet. Coupled w-lines, connectors, package models or s-parameter blocks can then be used in the widebus schematic to connect up each widebus group. One of the instances of a widebus group will be designated as the victim net on the schematic.

Widebus analysis can be performed on any project, even previous incarnations of a design. The following requirements are:

- One or more reference transfer nets (sheets) exist. The designator grouping on each unique transfer net are referred to as a “widebus group”.
- Each transfer nets consist of a single driver per transfer (they can have multiple transfers (i.e. memory read/memory write), and can have multiple receivers/loads.
- Transfer net characteristics such as transfers, UI, type, probe points and jitter are defined in the transfer net characteristics (not on a widebus sheet).

To create a widebus sheet there must be at least one transfer net included in the schematic set. Select **File > Schematic Sheet > New Sheet** from the app toolbar. In the newly opened New Sheet dialog box, select **Widebus** and select the widebus groups to be added to the sheet and the number of instances of each.

### **Clock Forwarding**

The **Parallel Link Designer** app supports the clock forwarding methodology defined in IBIS BIRD 204. This BIRD describes a mechanism by which the clock waveform, or recovered clock times, can be passed to the data AMI receiver model to allow the model maker access to the clock. This enables the receiver model to include clock phase noise, clock tree delays, phase interpolators, etc. IBIS BIRD204 has been approved by the IBIS committee and will be included in the IBIS 7.1.

A serial-link channel uses a CDR (clock data recovery) circuit in the receiver to create a clock signal that is then used to latch each individual data bit coming into the core of an IC. The IBIS-AMI standard was created to model this operation and a CDR is required to be present, either in the receiver model itself (the norm) or in the EDA tool that is running the simulation. By contrast, in a parallel-link channel no CDR is present. Instead, a separate clock (or strobe) signal is sent along with the data signals, which is used to latch sets of data bits at one time. This is known as clock-forwarding.

Clock forwarding is accessed by selection of a clock group in the Widebus Groups dialog box. The clock group identifies the clock and which respective data signals are associated with it. This relationship is used for analyzing clock recovery in the parallel interface.

Any widebus group transfer net that has been set up as a clock or strobe will be considered to be a clock group on the widebus sheet. The clock group drop down menu selection will show all strobe/clock groups on the current widebus sheet, plus “<none>”. Clock transfer nets will default to “<none>”. Data transfer nets will default to first clock or strobe associated sheet. Each data group can be associated with a different clock group which also supports sweeping of victim group.



## **See Also**

**Apps**  
**SI Viewer**

**Topics**  
“Analyze Parallel Links with Parallel Link Designer”  
“Configure DDR Controller with Two Memory Designators”

**Introduced in R2021b**

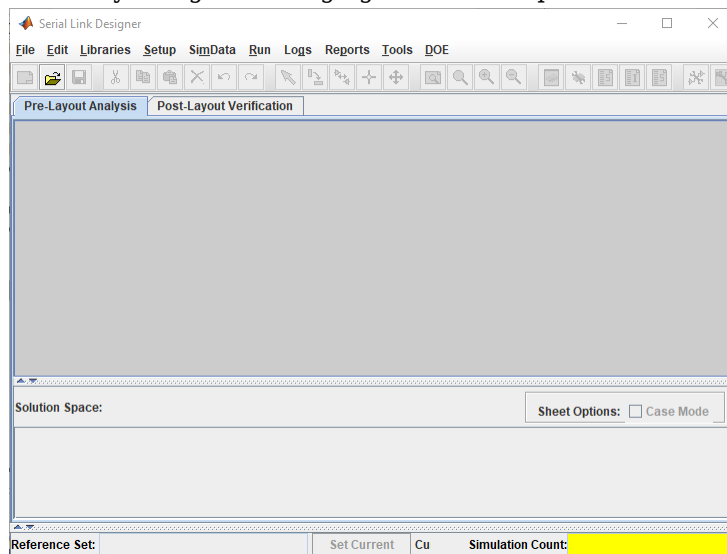
# Serial Link Designer

Analyze PCB designs for serial link applications

## Description

The **Serial Link Designer** app analyzes PCB designs for serial link applications for both pre-layout and post-layout simulations. To perform post-layout analysis, you need license to RF PCB Toolbox.

Using pre-layout analysis, you can run simulations using IBIS-AMI models with models of your system channels to determine equalization gamut, plot IBIS Statistical and Time Domain eye diagrams at a specified BER, measure channel response characteristics vs. specification masks, as well as perform crosstalk analysis. The pre-layout analysis environment can help you to generate design guidelines for board layouts, package layouts, connectors, and cabling. The post-layout environment verifies the actual layout against design guidelines or specification rules provided in a signal integrity kit.



## Open the Serial Link Designer App

- MATLAB Toolstrip: In the **Apps** tab, under **Signal Processing and Communications**, click the app icon.
- MATLAB command prompt: Enter `serialLinkDesigner`.

## Examples

- “Analyze Serial Links with Serial Link Designer”
- “Analyze Backplane with Line Cards”
- “Edit Imported S-Parameter Data”

## Programmatic Use

`serialLinkDesigner` opens a new session of the **Serial Link Designer** app, enabling you to design and analyze a serial link PCB.

`serialLinkDesigner('file.qcd')` opens the interface designated by `file.qcd`.

`serialLinkDesigner('file.script')` runs a script file and returns the app process handle.

## More About

### Simulation Parameters

Simulation Parameters control how the analysis is run in the **Serial Link Designer** app. For more information, see “Simulation Parameters Used in Serial Link Design”.

### Corner Conditions

There are two types of corner conditions you can vary using the **Serial Link Designer** app.

- IC Environment Corners — These are the temperature and voltage factor parameters that can be set for each IC corner and used in simulation. The temperature parameter has no affect on IBIS buffer models because those are pre-generated by the vendor and are already included in the model.
- Etch Corners — These are the scaling factors for the characteristic impedance ( $Z_0$ ) and propagation delay ( $T_{pd}$ ) parameters of transmission line models. Both ideal and lossy transmission line models are scaled. Lossy transmission line models are scaled by computing the values of  $Z_0$  and  $T_{pd}$  from the typical corner L and C values.

For more information, see “Specify Corner Conditions in Serial Link Design”.

### Stimulus Patterns

You can specify the stimulus patterns used by the simulator for time domain analysis. If the specified pattern for a designator has fewer bits than the simulation length, the pattern is repeated from the first bit of the pattern. If the pattern is longer than the simulation length, the simulation will end at the time specified by **Time Domain Stop**. For more information, see “Stimulus Patterns in Serial Link Design”.

## See Also

### Apps SI Viewer

### Topics

“Analyze Serial Links with Serial Link Designer”

“Analyze Backplane with Line Cards”

“Edit Imported S-Parameter Data”

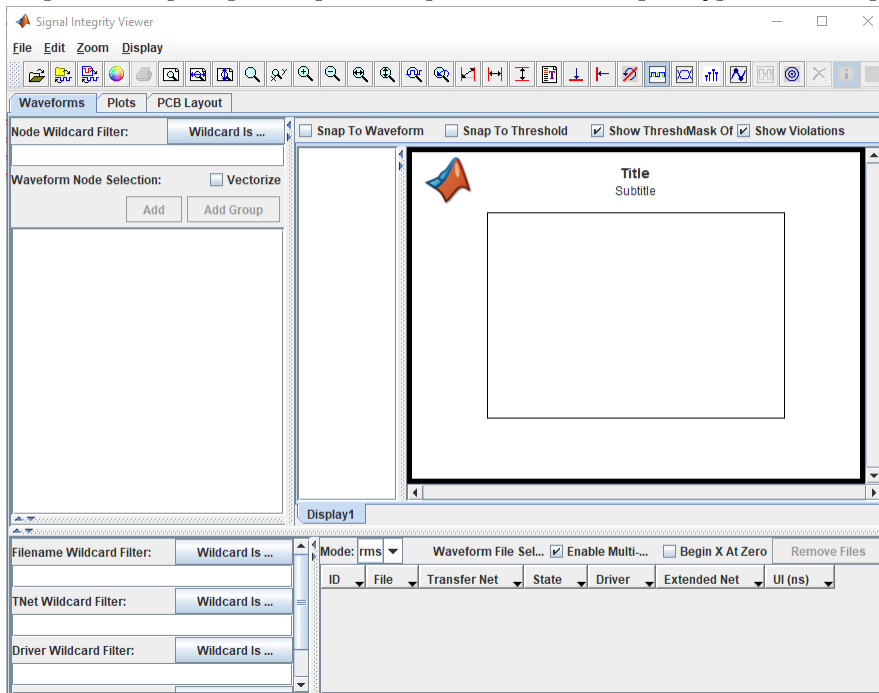
### Introduced in R2021b

# Signal Integrity Viewer

View the signal integrity results of Serial Link Designer or Parallel Link Designer app

## Description

Use the **Signal Integrity Viewer** app to view the results of the simulations done in **Serial Link Designer** or **Parallel Link Designer** app. This app provides a fast way to view waveforms, eye diagrams, TDR (time domain reflectometry)/ TDT (time domain transmission), s-parameters, impulse response, step response, pulse response, and other plot types for data produced by the two apps.



## Open the Signal Integrity Viewer App

- MATLAB Toolstrip: In the **Apps** tab, under **Signal Processing and Communications**, click the app icon.
- MATLAB command prompt: Enter `signalIntegrityViewer`.

## Examples

- “Analyze Serial Links with Serial Link Designer”
- “Analyze Parallel Links with Parallel Link Designer”

## Programmatic Use

`signalIntegrityViewer` opens a new session of the **Signal Integrity Viewer** app, enabling you to view the results of a serial link or parallel link analysis.

`signalIntegrityViewer('file.csv')` opens the waveform files. Waveform files can be `.csd` from IsSpice or `.tr0` files from HSPICE.

`signalIntegrityViewer('file.cfg')` opens a saved configuration file.

## More About

### See Also

#### Apps

**Serial Link Designer** | **Parallel Link Designer**

#### Topics

“Analyze Serial Links with Serial Link Designer”

“Analyze Parallel Links with Parallel Link Designer”

#### Introduced in R2021b

