sptool

Open interactive digital signal processing tool

Syntax

```
sptool
s = sptool('Signals')
f = sptool('Filters')
s = sptool('Spectra')
[s,ind] = sptool( __ )
s = sptool( __ ,0)
struc = sptool('create',paramlist)
sptool('load',struc)
struc = sptool('load',paramlist)
```

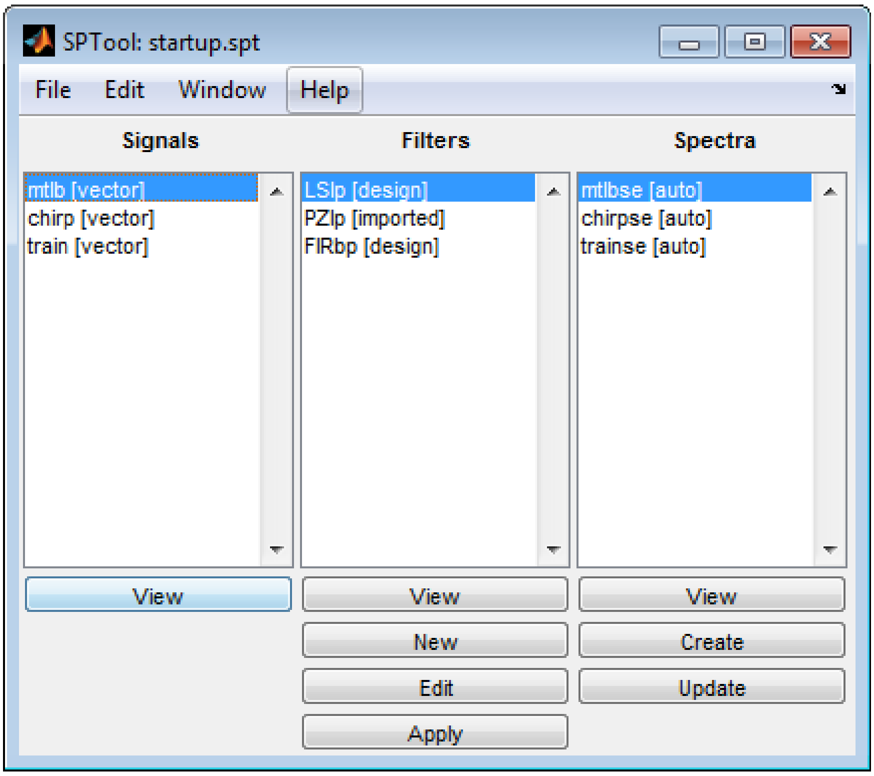
Description

Note

SPTool will be removed in a future release.

- For signal and spectral analysis, use the **Signal Analyzer** app instead.
- For filter design, use the **Filter Designer** app instead.
- For filter visualization, use `fvtool` instead.

The command, `sptool`, opens SPTool, a suite of four tools: Signal Browser, Filter Design and Analysis Tool, FVTool, and Spectrum Viewer. These tools provide access to many of the signal, filter, and spectral analysis functions in the toolbox. When you type `sptool` at the command line, the SPTool suite opens.



Using SPTool, you can:

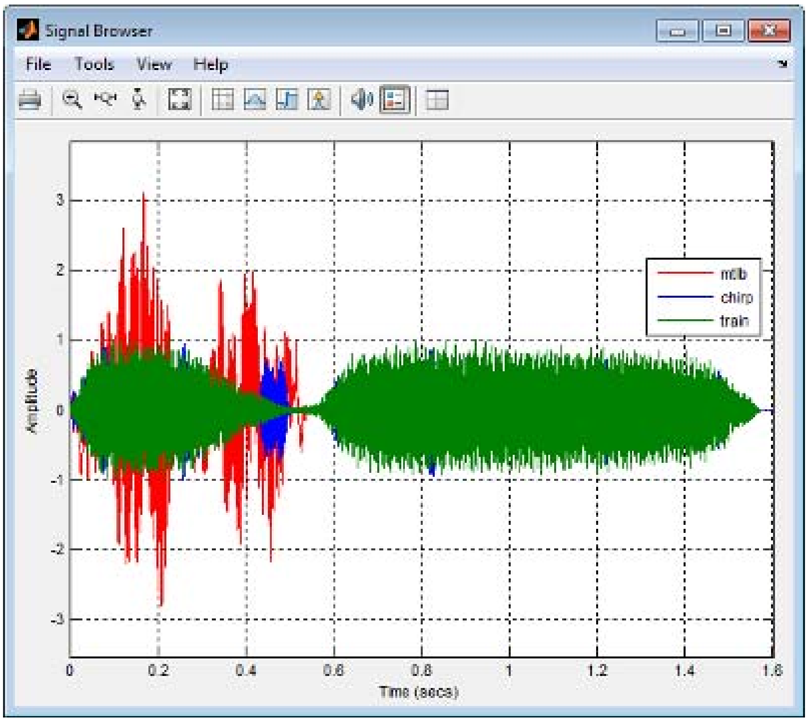
- Analyze signals listed in the **Signals** list box with the Signal Browser.
- Design or edit filters with the Filter Design and Analysis Tool (includes a Pole/Zero Editor).
- Analyze filter responses for filters listed in the **Filters** list box with FVTool.
- Apply filters in the **Filters** list box to signals in the **Signals** list box.
- Create and analyze signal spectra with the Spectrum Viewer.
- Print the Signal Browser, Filter Design and Analysis Tool, and Spectrum Viewer.

You can activate all four integrated signal processing tools from SPTool.

- Signal Browser
- Filter Designer App
- Filter Visualization Tool
- Spectrum Viewer

Signal Browser

The Signal Browser, hereafter referred to as the scope, allows you to view, measure, and analyze the time-domain information of one or more signals. To activate the Signal Browser, press the **View** button under the **Signals** list box in SPTool.



See the following sections for more information on the Signal Browser:

- Displaying Multiple Signals
- Signal Display
- Toolbar
- Measurements Panels
- Visuals — Time Domain Options
- Style Dialog Box

Displaying Multiple Signals

Multiple Signal Input

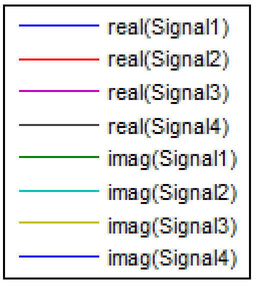
Select more than one signal in the **Signals** list box to show multiple signals within the same display or on separate displays. By default, the signals appear as different-colored lines on the same display. The signals can have different dimensions, sample rates, and data types. Each signal can be either real or complex valued.

Multiple Signal Colors

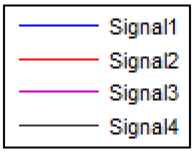
By default, Signal Browser has a white axes background and chooses line colors for each channel in a manner similar to the MATLAB® `plot` function. Signal Browser considers each of the real and imaginary components of the input signals to be a channel, and assigns each channel a line color in the following order:

1. Blue
2. Dark Green
3. Red
4. Cyan
5. Purple
6. Dark Yellow
7. Black

If there are more than 7 channels, the scope repeats this order to assign line colors to the remaining channels. For example, if you select 4 complex-valued input signals, the following legend appears in the display.



If all the input signals are real-valued, Signal Browser skips the line colors that would be associated with their imaginary components. For example, if you select 4 real-valued input signals, the following legend appears in the display.

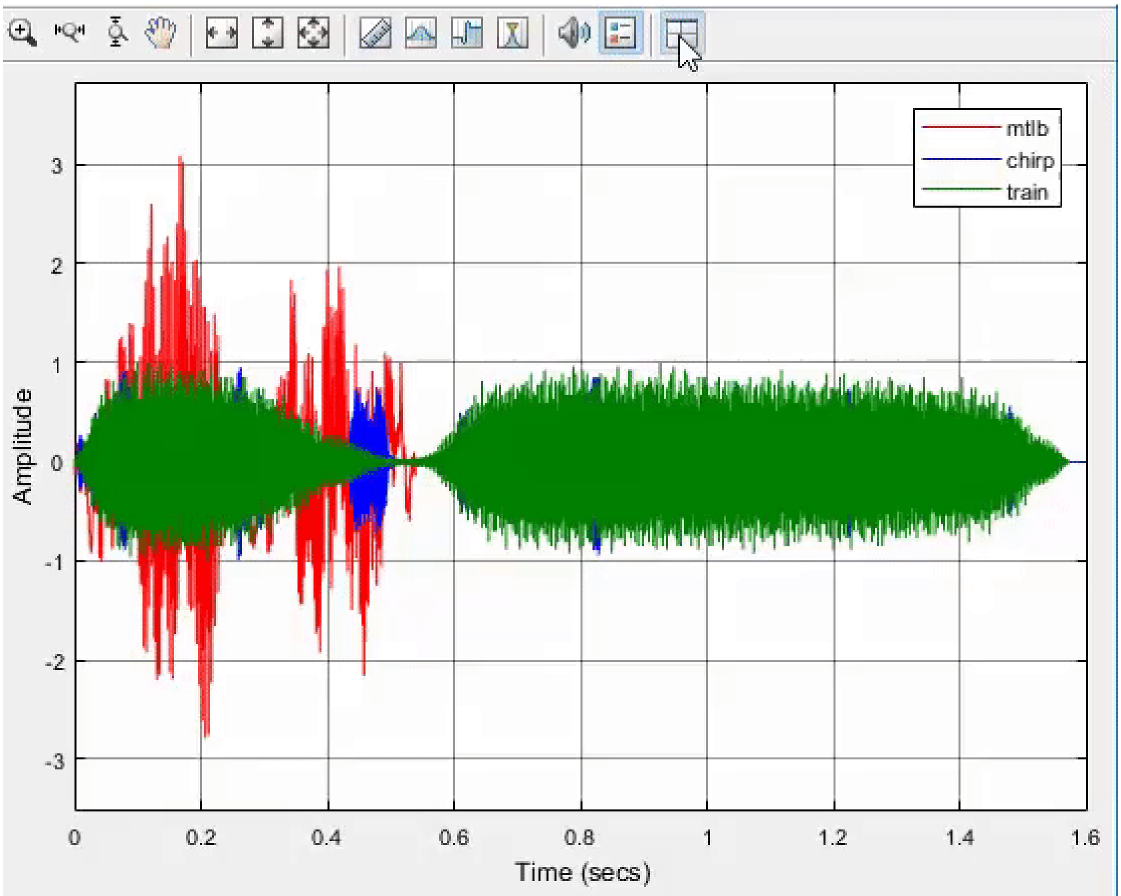


To manually modify any line color, select **View** > **Style** to open the Style dialog box. Next to **Properties for line**, select the signal name whose color you want to change. Then, next to **Line**, click the Line color button ( ) and select any color from the palette. To change the axes background color, click the Axes background color button ( ), and select any color from the palette.

Use Multiple Displays

You can display multiple channels of data on different displays in the window. In the toolbar, select **View** > **Layout**, or select the Layout button ( ).

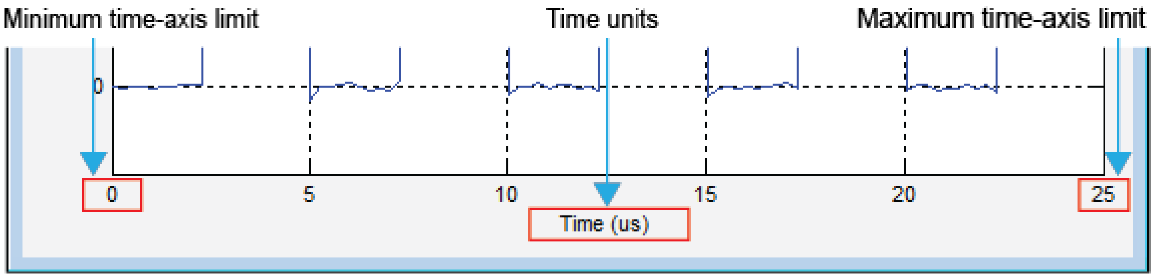
You can tile the window into multiple displays. For example, if there are three inputs to the tool, you can display the signals in three separate displays. The layout grid shows a 4 by 4 grid, but you can select up to 16 by 16 by clicking and dragging within the layout grid.



When you use the Layout option to tile the window into multiple displays, the display highlighted in blue is referred to as the *active display*. The dialog boxes reference the active display.

Signal Display

The Signal Browser uses the longest time length of all the input signals selected in the **Signals** list box for the time range. To communicate the array of times that corresponds to the current display, the scope uses the **Minimum time-axis limit**, **Time units**, and **Maximum time-axis limit** indicators on the scope window. The following figure highlights these aspects of the Signal Browser window.



- **Minimum time-axis limit** — The Signal Browser sets the minimum *time*-axis limit to 0.
- **Maximum time-axis limit** — The Signal Browser sets the maximum *time*-axis limit to the final time step of the longest input signal.
- **Time units** — The units used to describe the *time*-axis. The Signal Browser sets the time units using the value of the **Time Units** parameter on the **Main** tab of the Visuals:Time Domain Options dialog box. By default, this parameter is set to `Metric (based on Time Span)` and displays in metric units such as microseconds, milliseconds, minutes, days, etc. You can change the unit of measure to `Seconds` to always display the *time*-axis values in units of seconds. You can change it to `None` to suppress the display of units of measure on the *time*-axis. When you set this parameter to `None`, then the Signal Browser shows only the word `Time` on the *time*-axis.

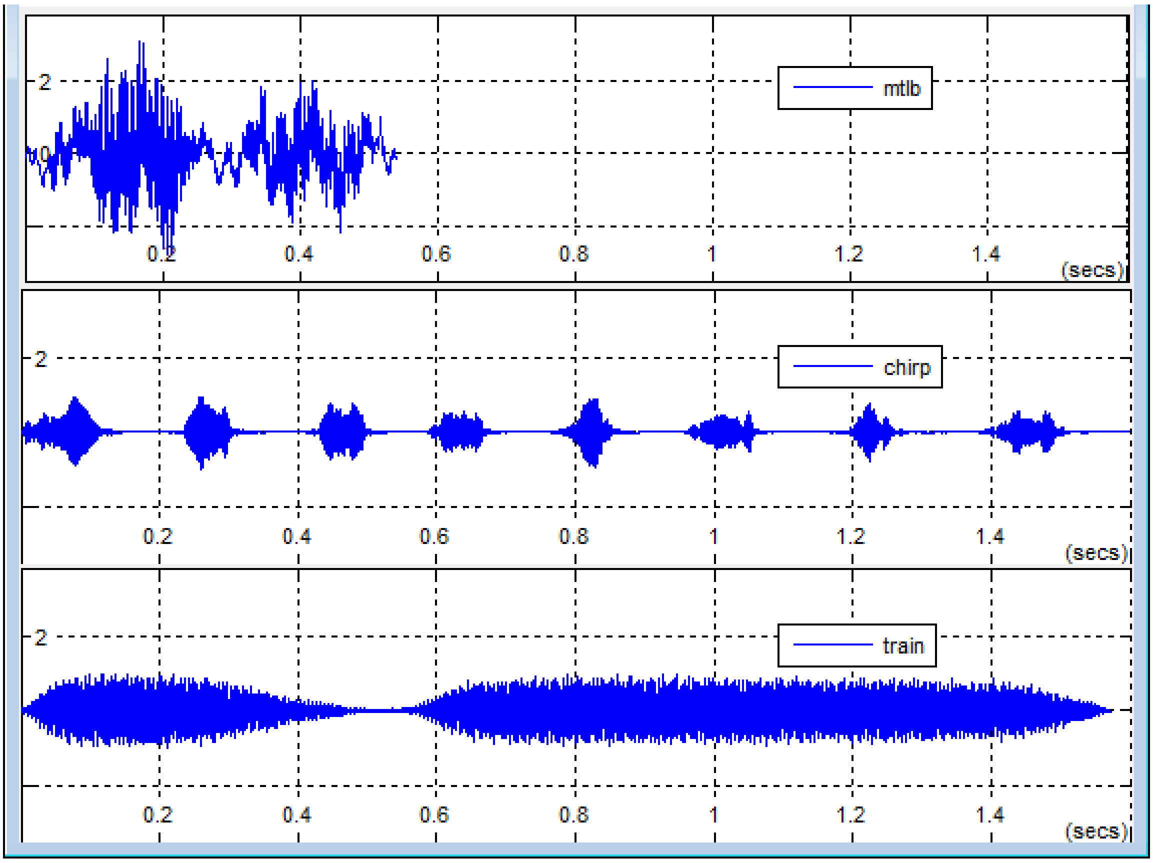
  To hide both the word `Time` and the values on the *time*-axis, set the **Show time-axis labels** parameter to `None`. To hide both the word `Time` and the values on the *time*-axis in all displays except the bottom ones in each column of displays, set this parameter to `Bottom Displays Only`. This behavior differs from that of the Simulink® Scope block, which always shows the values but never shows a label on the *x*-axis.

Signal Names and Legend Text

Signal Browser uses the names of the signals in the SPTool as the text displayed in the legends. If you change the name of any selected signal in the **Signals** list box, its corresponding legend entry in Signal Browser changes immediately. To change the name of any selected signal, from the SPTool menu select **Edit** > **Name**. Signal Browser automatically updates the legend to reflect the new signal name you entered. Similarly, if you modify any entry in a legend in Signal Browser, then SPTool updates the corresponding signal name in the **Signals** list box.

Axes Maximization

You can specify whether to display the Signal Browser in maximized axes mode. In this mode, the axes are expanded to fill the entire display. In each display, there is no space to show titles or axis labels. The minimum and maximum *time*-axis limits are located at the far-left and far-right edges of the display. The values at the axis tick marks appear as grid lines on top of the axes. The following figure highlights how three displays appear in maximized axes mode in the Signal Browser window.



To enable or disable this mode, in the Signal Browser menu, select **View** > **Properties** to bring up the Visuals:Time Domain Options dialog box. In the **Main** pane, you can set the **Maximize axes** parameter to one of the following options:

- `Auto` — In this mode, the axes appear maximized in all displays only if the **Title** and **Y-Axis label** parameters are empty for every display. If you enter any value in any display for either of these parameters, the axes are not maximized.
- `On` — In this mode, the axes appear maximized in all displays. Any values entered into the **Title** and **Y-Axis label** parameters are hidden.
- `Off` — In this mode, none of the axes appear maximized.

See the Visuals — Time Domain Options section for more information.

Toolbar

The Signal Browser toolbar contains the following buttons.

Print Button

Button	Menu Location	Shortcut Keys	Description
	**File** > **Print**	**Ctrl+P**	Print the current scope window. To print the current scope window to a figure rather than sending it to your printer, select **File** > **Print to figure**.

Zoom and Axes Control Buttons

Button	Menu Location	Shortcut Keys	Description
	**Tools** > **Zoom In**	N/A	When this tool is active, you can zoom in on the scope window. To do so, click in the center of your area of interest, or click and drag your cursor to draw a rectangular area of interest inside the scope window.

Button	Menu Location	Shortcut Keys	Description
	**Tools** > **Zoom X**	N/A	You access the Zoom X button from the menu under the Zoom In icon. When this tool is active, you can zoom in on the x-axis. To do so, click inside the scope window, or click and drag your cursor along the x-axis over your area of interest.
	**Tools** > **Zoom Y**	N/A	You access the Zoom Y button from the menu under the Zoom In icon. When this tool is active, you can zoom in on the y-axis. To do so, click inside the scope window, or click and drag your cursor along the y-axis over your area of interest.
	**Tools** > **Pan**	N/A	You access the Pan button from the menu under the Zoom In icon. When this tool is active, you can pan on the scope window. To do so, click in the center of your area of interest and drag your cursor to the left, right, up, or down, to move the position of the display.
	**Tools** > **Scale Y-Axis Limits**	**Ctrl+A**	Click this button to scale the axes in the active scope window. Alternatively, you can enable automatic axes scaling by selecting one of the following options from the **Tools** menu: • **Automatically Scale Axes Limits** — When you select this option, the scope scales the axes as needed during simulation. • **Scale Axes Limits after 10 Updates** — When you select this option, the scope scales the axes after 10 updates. The scope does not scale the axes again during the simulation. • **Scale Axes Limits at Stop** — When you select this option, the scope scales the axes each time the simulation is stopped.
	**Tools** > **Scale X-Axis Limits**	N/A	You access the Scale X-Axis Limits button from the menu under the current Axis Limits icon. Click this button to scale the axes in the X direction in the active scope window. Alternatively, you can enable automatic axes scaling by selecting one of the following options from the **Tools** menu: • **Automatically Scale Axes Limits** — When you select this option, the scope scales the axes as needed during simulation. • **Scale Axes Limits after 10 Updates** — When you select this option, the scope scales the axes after 10 updates. The scope does not scale the axes again during the simulation. • **Scale Axes Limits at Stop** — When you select this option, the scope scales the axes each time the simulation is stopped.
	**Tools** > **Scale X & Y Axes Limits**	N/A	You access the Scale X & Y Axes Limits button from the menu under the current Axis Limits icon. Click this button to scale the axes in both the X and Y directions in the active scope window. Alternatively, you can enable automatic axes scaling by selecting one of the following options from the **Tools** menu: • **Automatically Scale Axes Limits** — When you select this option, the scope scales the axes as needed during simulation. • **Scale Axes Limits after 10 Updates** — When you select this option, the scope scales the axes after 10 updates. The scope does not scale the axes again during the simulation. • **Scale Axes Limits at Stop** — When you select this option, the scope scales the axes each time the simulation is stopped.

Measurements Buttons

	Tools > **Measurements** > **Cursor Measurements**	N/A	Open or close the **Cursor Measurements** panel. This panel puts screen cursors on all the displays. See the Cursor Measurements Panel section for more information.
	Tools > **Measurements** > **Signal Statistics**	N/A	Open or close the **Signal Statistics** panel. This panel displays the maximum, minimum, peak-to-peak difference, mean, median, RMS values of a selected signal, and the times at which the maximum and minimum occur. See the Signal Statistics Panel section for more information.
	Tools > **Measurements** > **Bilevel Measurements**	N/A	Open or close the **Bilevel Measurements** panel. This panel displays information about a selected signal's transitions, overshoots or undershoots, and cycles. See the Bilevel Measurements Panel section for more information.
	Tools > **Measurements** > **Peak Finder**	N/A	Open or close the **Peak Finder** panel. This panel displays maxima and the times at which they occur, allowing the settings for peak threshold, maximum number of peaks, and peak excursion to be modified. See the Peak Finder Panel section for more information.

Other Buttons

	Tools > **Play Selected Signal**	N/A	Play an audio signal. The function `soundsc` is used to play the signal.
	View > **Show All Legends**	N/A	Show a legend that matches each line style to a signal name in every display.
	View > **Layout**	N/A	Arrange the layout of displays in the Signal Browser. This feature allows you to tile your screen into a number of separate displays, up to a grid of 4 rows and 4 columns. You may find multiple displays useful when you select multiple input signals in SPTool. The default display is 1 row and 1 column.

You can control whether this toolbar appears in the Signal Browser window. From the Signal Browser menu, select **View** > **Toolbar**.

Measurements Panels

The Measurements panels are the five panels that appear at the right side of the Signal Browser. These panels are labeled **Trace selection**, **Cursor measurements**, **Signal statistics**, **Bilevel measurements**, and **Peak finder**.

Measurements Panel Buttons

Each of the Measurements panels contains the following buttons that enable you to modify the appearance of the current panel.

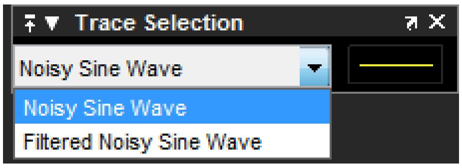
Button	Description
	Move the current panel to the top. When you are displaying more than one panel, this action moves the current panel above all the other panels.
	Collapse the current panel. When you first enable a panel, by default, it displays one or more of its panes. Click this button to hide all of its panes to conserve space. After you click this button, it becomes the expand button .
	Expand the current panel. This button appears after you click the collapse button to hide the panes in the current panel. Click this button to display the panes in the current panel and show measurements again. After you click this button, it becomes the collapse button again.

Button	Description
	Undock the current panel. This button lets you move the current panel into a separate window that can be relocated anywhere on your screen. After you click this button, it becomes the dock button in the new window.
	Dock the current panel. This button appears only after you click the undock button. Click this button to put the current panel back into the right side of the Scope window. After you click this button, it becomes the undock button again.
	Close the current panel. This button lets you remove the current panel from the right side of the Scope window.

Some panels have their measurements separated by category into a number of panes. Click the pane expand button ▶ to show each pane that is hidden in the current panel. Click the pane collapse button ▼ to hide each pane that is shown in the current panel.

Trace Selection Panel

When you use the scope to view multiple signals, the Trace Selection panel appears if you have more than one signal displayed and you click any of the other Measurements panels. The Measurements panels display information about only the signal chosen in this panel. Choose the signal name for which you would like to display time domain measurements. See the following figure.



You can choose to hide or display the **Trace Selection** panel. In the Scope menu, select **Tools** > **Measurements** > **Trace Selection**.

Cursor Measurements Panel

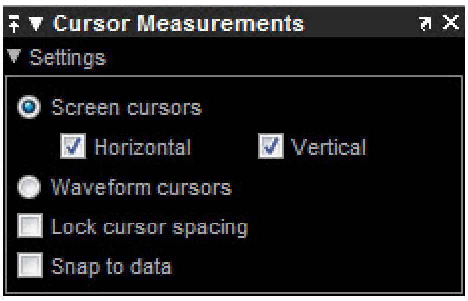
The **Cursor Measurements** panel displays screen cursors. The panel provides two types of cursors for measuring signals. Waveform cursors are vertical cursors that track along the signal. Screen cursors are both horizontal and vertical cursors that you can place anywhere in the display.

Note

If a data point in your signal has more than one value, the cursor measurement at that point is undefined and no cursor value is displayed.

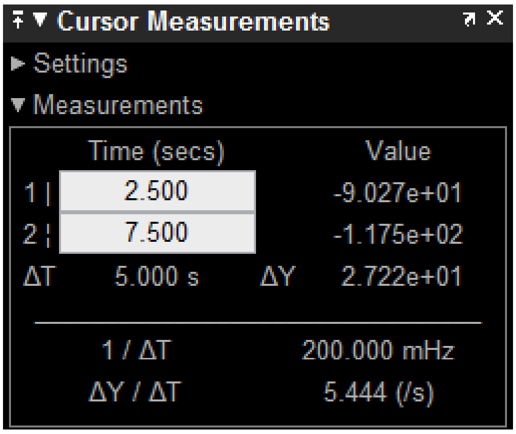
In the Scope menu, select **Tools** > **Measurements** > **Cursor Measurements**. Alternatively, in the Scope toolbar, click the Cursor Measurements button.

The **Settings** pane enables you to modify the type of screen cursors used for calculating measurements. When more than one signal is displayed, you can assign cursors to each trace individually.



- **Screen Cursors** — Shows screen cursors.
- **Horizontal** — Shows horizontal screen cursors.
- **Vertical** — Shows vertical screen cursors.

- **Waveform Cursors** — Shows cursors that attach to the input signals.
- **Lock Cursor Spacing** — Locks the frequency difference between the two cursors.
- **Snap to Data** — Positions the cursors on signal data points.



You can use the mouse or the left and right arrow keys to move vertical or waveform cursors and the up and down arrow keys for horizontal cursors.

The **Measurements** pane shows the time and value measurements.

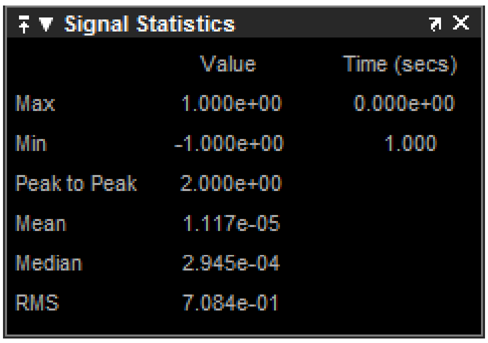
- **1 |**— Shows or enables you to modify the time or value at cursor number one, or both.
- **2 :**— Shows or enables you to modify the time or value at cursor number two, or both.
- **Δt**— Shows the absolute value of the difference in the times between cursor number one and cursor number two.
- **ΔV**— Shows the absolute value of the difference in signal amplitudes between cursor number one and cursor number two.
- **1/Δt**— Shows the rate, the reciprocal of the absolute value of the difference in the times between cursor number one and cursor number two.
- **ΔV/Δt**— Shows the scope, the ratio of the absolute value of the difference in signal amplitudes between cursors to the absolute value of the difference in the times between cursors.

Signal Statistics Panel

Note

The **Signal Statistics** panel requires a DSP System Toolbox™ or Simscape™ license.

The **Signal Statistics** panel displays the maximum, minimum, peak-to-peak difference, mean, median, and RMS values of a selected signal. It also shows the x-axis indices at which the maximum and minimum values occur. In the Scope menu, select **Tools** > **Measurements** > **Signal Statistics**. Alternatively, in the scope toolbar, click the Signal Statistics button.



The statistics shown are:

- **Max** — The maximum or largest value within the displayed portion of the input signal. For more information on the algorithm this measurement uses, see the MATLAB `max` function reference.

- **Min** — The minimum or smallest value within the displayed portion of the input signal. For more information on the algorithm this measurement uses, see the MATLAB `min` function reference.
- **Peak to Peak** — The difference between the maximum and minimum values within the displayed portion of the input signal. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox™ `peak2peak` function reference.
- **Mean** —The average or mean of all the values within the displayed portion of the input signal. For more information on the algorithm this measurement uses, see the MATLAB `mean` function reference.
- **Median** — The median value within the displayed portion of the input signal. For more information on the algorithm this measurement uses, see the MATLAB `median` function reference.
- **RMS** — Shows the difference between the maximum and minimum values within the displayed portion of the input signal. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `rms` function reference.

When you use the zoom options in the Scope, the Signal Statistics measurements automatically adjust to the time range shown in the display. For example, you can zoom in on one pulse to make the **Signal Statistics** panel display information about only that particular pulse.

The Signal Statistics measurements are valid for any units of the input signal. The letter after the value associated with each measurement represents the appropriate International System of Units (SI) prefix, such as *m* for *milli-*. For example, if the input signal is measured in volts, an *m* next to a measurement value indicates that this value is in units of millivolts.

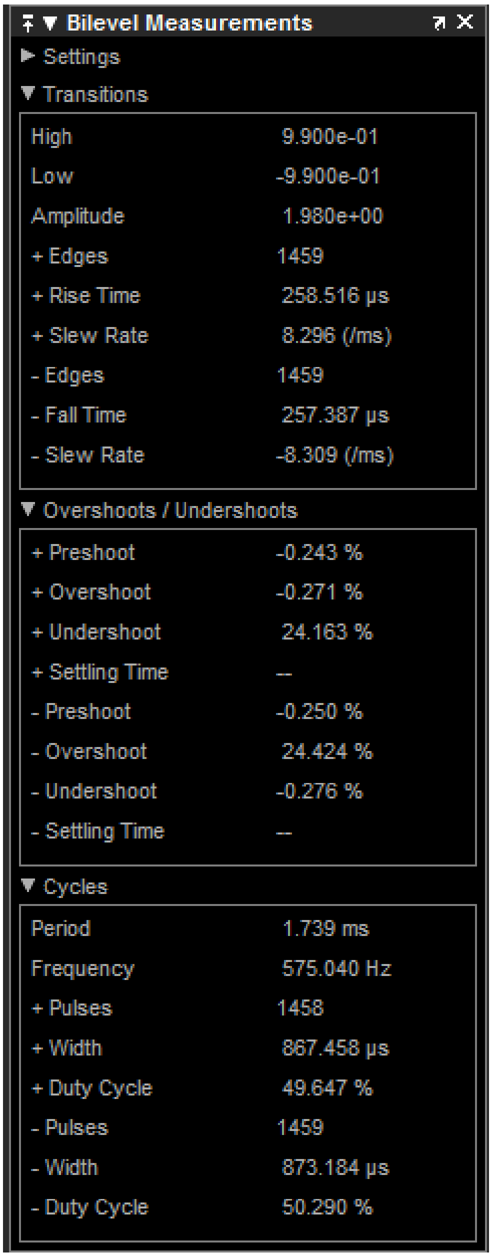
Bilevel Measurements Panel

Note

The **Bilevel Measurements** panel requires a DSP System Toolbox or Simscape license.

The **Bilevel Measurements** panel shows information about transitions, overshoots, undershoots, and cycles for a selected signal. You can choose to hide or display the **Bilevel Measurements** panel. In the scope menu, select **Tools** > **Measurements** > **Bilevel Measurements**. Alternatively, in the scope toolbar, you can select the Bilevel Measurements button.

When you use the zoom options in the Scope, the bilevel measurements automatically adjust to the time range shown in the display. For example, you can zoom in on one rising edge to make the **Bilevel Measurements** panel display information about only that particular rising edge. This feature does not apply to the **High** and **Low** measurements.



The **Bilevel Measurements** panel is separated into four panes, labeled **Settings**, **Transitions**, **Overshoots/Undershoots**, and **Cycles**. You can expand each pane to see the available options.

Bilevel Measurements Plot

Note

Opening the **Overshoots/Undershoots** pane may reduce the number of available edges by 1 if the last edge of the pulse is not within the **Settle Seek** time of the end of the plot. This reduction in the number of edges changes the values of **Edges**, **Fall Time**, and **Slew Rate** in the **Transitions** pane.

The **Settings** pane enables you to modify the properties used to calculate various measurements involving transitions, overshoots, undershoots, and cycles. You can modify the high-state level, low-state level, state-level tolerance, upper-reference level, mid-reference level, and lower-reference level, as shown in the following figure.

- **Auto State Level** — When this check box is selected, the Bilevel measurements panel autodetects the high- and low- state levels of a bilevel waveform. For more information on the algorithm this option uses, see the Signal Processing Toolbox `statelevels` function reference. When this check box is cleared, you can enter in values for the high- and low- state levels manually.
  - **High** — Manually specify the value for a positive polarity or high-state level.
  - **Low** — Manually specify the value for a negative polarity or low-state level.

- **State Level Tolerance** — Tolerance within which the initial and final levels of each transition must be within their respective state levels. This value is expressed as a percentage of the difference between the high- and low-state levels.
- **Upper Ref Level** — Used to compute the end of the rise-time measurement or the start of the fall time measurement. This value is expressed as a percentage of the difference between the high- and low-state levels.
- **Mid Ref Level** — Used to determine when a transition occurs. This value is expressed as a percentage of the difference between the high- and low- state levels. The mid-reference level is shown as a horizontal line, and its corresponding mid-reference level instant is shown as a vertical line.
- **Lower Ref Level** — Used to compute the end of the fall-time measurement or the start of the rise-time measurement. This value is expressed as a percentage of the difference between the high- and low-state levels.
- **Settle Seek** — The duration after the mid-reference level instant when each transition occurs used for computing a valid settling time. This value is equivalent to the input parameter, `D`, which you can set when you run the `settlingtime` function. The settling time is displayed in the **Overshoots/Undershoots** pane.

The **Transitions** pane displays calculated measurements associated with the input signal changing between its two possible state level values, high and low. The Transition measurements assume that the amplitude of the input signal is in units of volts. Convert all input signals to volts for the Transition measurements to be valid.

A positive-going transition, or *rising edge*, in a bilevel waveform is a transition from the low-state level to the high-state level. A positive-going transition has a slope value greater than zero. Whenever there is a plus sign (+) next to a text label, this symbol refers to measurement associated with a rising edge, a transition from a low-state level to a high-state level.

A negative-going transition, or falling edge, in a bilevel waveform is a transition from the high-state level to the low-state level. A negative-going transition has a slope value less than zero. Whenever there is a minus sign (–) next to a text label, this symbol refers to measurement associated with a falling edge, a transition from a high-state level to a low-state level.

- **High** — The high-amplitude state level of the input signal over the duration of the **Time Span** parameter. You can set **Time Span** in the **Main** or**Time** tab of the Visuals—Time Domain Properties dialog box. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `statelevels` function reference.
- **Low** — The low-amplitude state level of the input signal over the duration of the **Time Span** parameter. You can set **Time Span** in the **Main** or **Time** tab of the Visuals—Time Domain Properties dialog box. The tab in which **Time Span** appears depends on whether you launched the scope from a from a MATLAB System object™ or from a Simulink block, respectively. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `statelevels` function reference.
- **Amplitude** — Difference in amplitude between the high-state level and the low-state level.
- **+ Edges** — Total number of positive-polarity, or rising, edges counted within the displayed portion of the input signal.
- **+ Rise Time** — Average amount of time required for each rising edge to cross from the lower-reference level to the upper-reference level. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `risetime` function reference.
- **+ Slew Rate** — Average slope of each rising-edge transition line within the upper- and lower-percent reference levels in the displayed portion of the input signal. The region in which the slew rate is calculated appears in gray.

  For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `slewrate` function reference.
- **– Edges** — Total number of negative-polarity or falling edges counted within the displayed portion of the input signal.
- **– Fall Time** — Average amount of time required for each falling edge to cross from the upper-reference level to the lower-reference level. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `falltime` function reference.
- **– Slew Rate** — Average slope of each falling edge transition line within the upper- and lower-percent reference levels in the displayed portion of the input signal. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `slewrate` function reference.

The **Overshoots/Undershoots** pane displays calculated measurements involving the distortion and damping of the input signal. *Overshoot* and *undershoot* refer to the amount that a signal, respectively, exceeds and falls below its final steady-state value. *Preshoot* refers to the amount before a transition that a signal varies from its initial steady-state value. This figure shows preshoot, overshoot, and undershoot for a rising-edge transition.

Overshoot/Undershoot Plot

- + *Preshoot* — Average lowest aberration in the region immediately preceding each rising transition.
- + *Overshoot* — Average highest aberration in the region immediately following each rising transition. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `overshoot` function reference.
- + *Undershoot* — Average lowest aberration in the region immediately following each rising transition. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `undershoot` function reference.
- + *Settling Time* — Average time required for each rising edge to enter and remain within the tolerance of the high-state level for the remainder of the settle seek duration. The settling time is the time after the mid-reference level instant when the signal crosses into and remains in the tolerance region around the high-state level. This crossing is illustrated in the following figure.

  Settling Time Plot

  You can modify the settle seek duration parameter in the **Settings** pane. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `settlingtime` function reference.
- – *Preshoot* — Average highest aberration in the region immediately preceding each falling transition.
- – *Overshoot* — Average highest aberration in the region immediately following each falling transition. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `overshoot` function reference.
- – *Undershoot* — Average lowest aberration in the region immediately following each falling transition. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `undershoot` function reference.
- – *Settling Time* — Average time required for each falling edge to enter and remain within the tolerance of the low-state level for the remainder of the settle seek duration. The settling time is the time after the mid-reference level instant when the signal crosses into and remains in the tolerance region around the low-state level. You can modify the settle seek duration parameter in the **Settings** pane. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `settlingtime` function reference.

The **Cycles** pane displays calculated measurements of to repetitions or trends in the displayed portion of the input signal.

- **Period** — Average duration between adjacent edges of identical polarity within the displayed portion of the input signal. The Bilevel measurements panel calculates period as follows. It takes the difference between the mid-reference level instants of the initial transition of each positive-polarity pulse and the next positive-going transition. These mid-reference level instants appear as red dots.

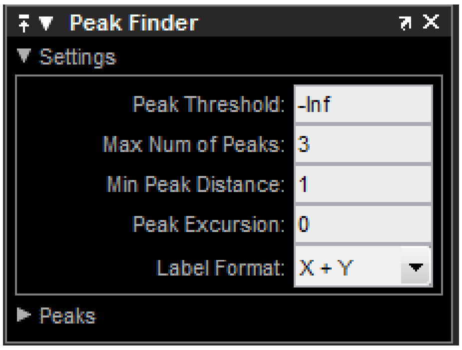
  For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `pulseperiod` function reference.
- **Frequency** — Reciprocal of the average period. Whereas period is typically measured in some metric form of seconds, or seconds per cycle, frequency is typically measured in hertz or cycles per second.
- **+ Pulses** — Number of positive-polarity pulses counted.
- **+ Width** — Average duration between rising and falling edges of each positive-polarity pulse within the displayed portion of the input signal. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `pulsewidth` function reference.
- **+ Duty Cycle** — Average ratio of pulse width to pulse period for each positive-polarity pulse within the displayed portion of the input signal. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `dutycycle` function reference.
- **– Pulses** — Number of negative-polarity pulses counted.
- **– Width** — Average duration between rising and falling edges of each negative-polarity pulse within the

displayed portion of the input signal. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `pulsewidth` function reference.

- **– Duty Cycle** — Average ratio of pulse width to pulse period for each negative-polarity pulse within the displayed portion of the input signal. For more information on the algorithm this measurement uses, see the Signal Processing Toolbox `dutycycle` function reference.

Peak Finder Panel

The **Peak Finder** panel displays the maxima, showing the *x*-axis values at which they occur. Peaks are defined as a local maximum where lower values are present on both sides of a peak. Endpoints are not considered to be peaks. This panel allows you to modify the settings for peak threshold, maximum number of peaks, and peak excursion. You can choose to hide or display the **Peak Finder** panel. In the scope menu, select **Tools** > **Measurements** > **Peak Finder**. Alternatively, in the scope toolbar, select the Peak Finder button.

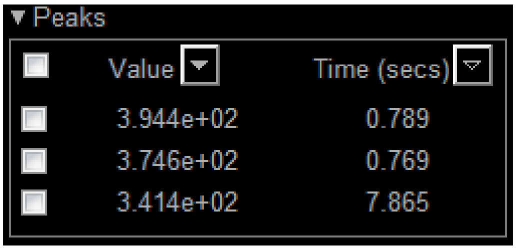


The **Peak finder** panel is separated into two panes, labeled **Settings** and **Peaks**. You can expand each pane to see the available options.

The **Settings** pane enables you to modify the parameters used to calculate the peak values within the displayed portion of the input signal. For more information on the algorithms this pane uses, see the Signal Processing Toolbox `findpeaks` function reference.

- **Peak Threshold** — The level above which peaks are detected. This setting is equivalent to the `MINPEAKHEIGHT` parameter, which you can set when you run the `findpeaks` function.
- **Max Num of Peaks** — The maximum number of peaks to show. The value you enter must be a scalar integer from 1 through 99. This setting is equivalent to the `NPEAKS` parameter, which you can set when you run the `findpeaks` function.
- **Min Peaks Distance** — The minimum number of samples between adjacent peaks. This setting is equivalent to the `MINPEAKDISTANCE` parameter, which you can set when you run the `findpeaks` function.
- **Peak Excursion** — The minimum height difference between a peak and its neighboring samples The peak excursion setting is equivalent to the `THRESHOLD` parameter, which you can set when you run the `findpeaks` function.
- **Label Format** — The coordinates to display next to the calculated peak values on the plot. To see peak values, expand the **Peaks** pane and select the check boxes associated with individual peaks of interest. By default, both *x*-axis and *y*-axis values are displayed on the plot. Select which axes values you want to display next to each peak symbol on the display.
  - X+Y — Display both *x*-axis and *y*-axis values.
  - X — Display only *x*-axis values.
  - Y — Display only *y*-axis values.

The **Peaks** pane displays all of the largest calculated peak values. It also shows the coordinates at which the peaks occur, using the parameters you define in the **Settings** pane. You set the **Max Num of Peaks** parameter to specify the number of peaks shown in the list.



The numerical values displayed in the **Value** column are equivalent to the `pks` output argument returned when you run the `findpeaks` function. The numerical values displayed in the second column are similar to the `locs` output argument returned when you run the `findpeaks` function.

The Peak Finder displays the peak values in the **Peaks** pane. By default, the **Peak Finder** panel displays the largest calculated peak values in the **Peaks** pane in decreasing order of peak height. Use the sort descending button ( ) to rearrange the category and order by which Peak Finder displays peak values. Click this button again to sort the peaks in ascending order instead. When you do so, the arrow changes direction to become the sort ascending button ( ). A filled sort button indicates that the peak values are currently sorted in the direction of the button arrow. If the sort button is not filled ( ), then the peak values are sorted in the opposite direction of the button arrow. The **Max Num of Peaks** parameter still controls the number of peaks listed.

Use the check boxes to control which peak values are shown on the display. By default, all check boxes are cleared and the **Peak Finder** panel hides all the peak values. To show all the peak values on the display, select the check box in the top-left corner of the **Peaks** pane. To hide all the peak values on the display, clear this check box. To show an individual peak, select the check box directly to the left of its **Value** listing. To hide an individual peak, clear the check box directly to the left of its **Value** listing.

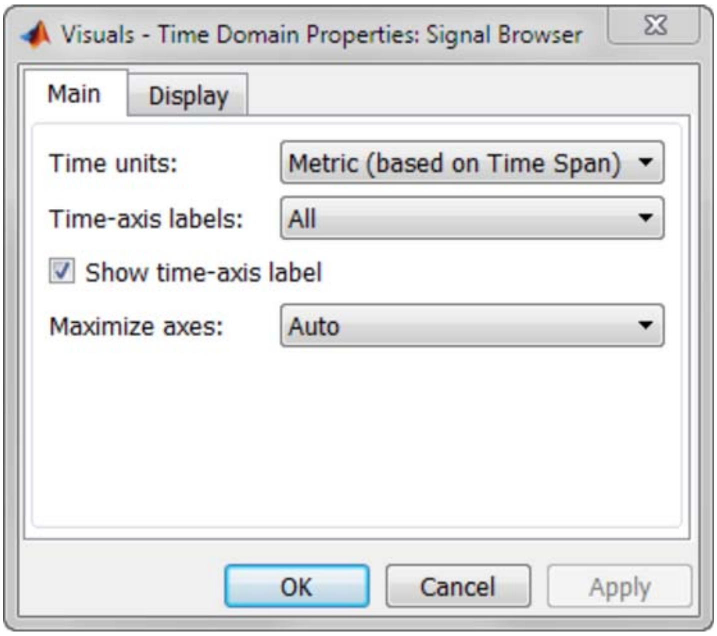
The Peaks are valid for any units of the input signal. The letter after the value associated with each measurement indicates the abbreviation for the appropriate International System of Units (SI) prefix, such as *m* for *milli-*. For example, if the input signal is measured in volts, an *m* next to a measurement value indicates that this value is in units of millivolts.

Visuals — Time Domain Options

The Visuals — Time Domain Properties dialog box controls the visual configuration settings of the Signal Browser display. From the menu, select **View** > **Configuration Properties** to open this dialog box.

Main Pane

The **Main** pane of the Visuals — Time Domain Properties dialog box appears as follows.



Time units

Specify the units used to describe the *time*-axis. The default setting is `Metric`. You can select one of the following options.

- `Metric` — In this mode, the Signal Browser converts the times on the *time*-axis to some metric units such as milliseconds, microseconds, days, etc. The Signal Browser chooses the appropriate metric units, based on the minimum *time*-axis limit and the maximum *time*-axis limit of the window.
- `Seconds` — In this mode, the Signal Browser always displays the units on the *time*-axis as seconds.
- `None` — In this mode, the Signal Browser displays no units on the *time*-axis. The Signal Browser shows only the word `Time` on the *time*-axis.

Time-axis labels

Specify how to display the time units used to describe the *time*-axis. The default setting is `All`. You can select one of the following options.

- `All` — In this mode, the *time*-axis labels appear in all displays.
- `None` — In this mode, the *time*-axis labels do not appear in the displays.
- `Bottom Displays Only` — In this mode, the *time*-axis labels appear only in the bottom row of the displays.

Show time-axis label

Select to turn on *time*-axis label display.

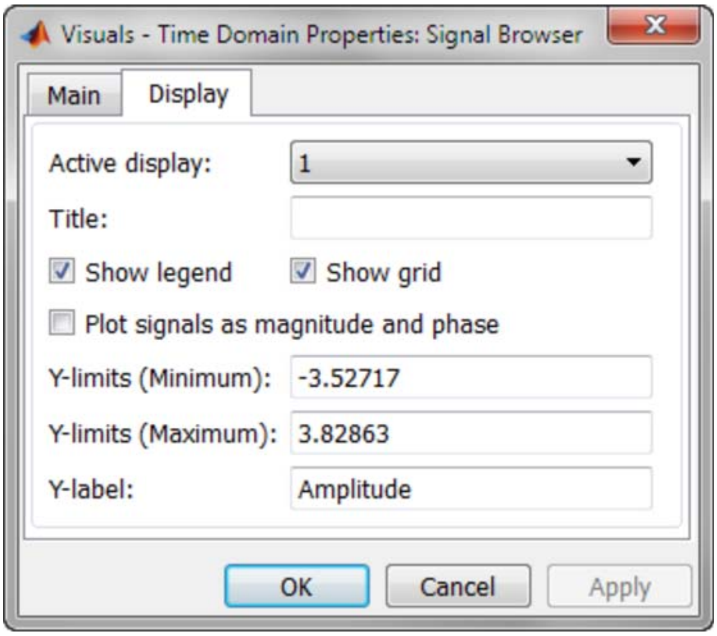
Maximize axes

Specify whether to display the Signal Browser in maximized axes mode. In this mode, each of the axes is expanded to fit into the entire display. In each display, there is no space to show labels. Tick mark values are shown on top of the plotted data. The default setting is `Auto`. You can select one of the following options:

- `Auto` — In this mode, the axes appear maximized in all displays only if the **Title** and **Y-Axis label** parameters are empty for every display. If you enter any value in any display for either of these parameters, the axes are not maximized.
- `On` — In this mode, the axes appear maximized in all displays. Any values entered into the **Title** and **Y-Axis label** parameters are hidden.
- `Off` — In this mode, none of the axes appear maximized.

Display Pane

The **Display** pane of the Visuals — Time Domain Properties dialog box appears as follows.



Active display

Specify the active display as an integer to get and set relevant properties. The number of a display corresponds to its column-wise placement index. Set this property to control which display has its axes colors, line properties, marker properties, and visibility changed.

When you use the Layout option to tile the window into multiple displays, the display highlighted in blue is referred to as the *active display*. The default setting is 1.

Title

Specify the active display title as text. By default, the active display has no title.

Show legend

Select this check box to show the legend in the display. The channel legend displays a name for each channel of each input signal. When the legend appears, you can place it anywhere inside of the scope window. To turn off the legend, clear the **Show legend** check box.

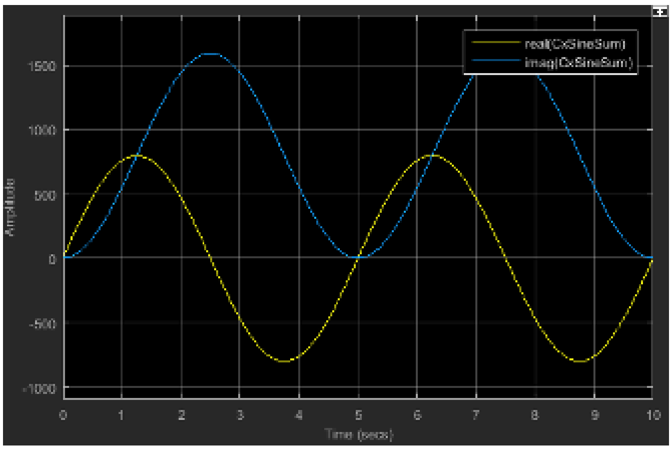
You can edit the name of any channel in the legend by double-clicking the current name and entering a new channel name. By default, if the signal has multiple channels, the scope uses an index number to identify each channel of that signal. To change the appearance of any channel of any input signal in the scope window, from the scope menu, select **View** > **Style**. The legend lets you modify what signals are shown. To show only one signal, click the signal name. To toggle a signal on/off, right-click the signal name.

Show grid

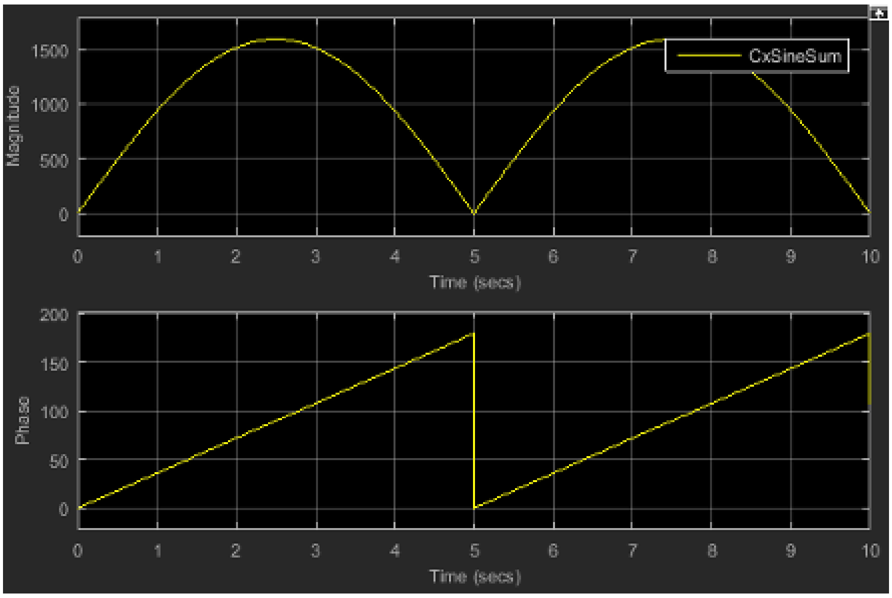
When you select this check box, a grid appears in the display of the scope figure. To hide the grid, clear this check box.

Plot signals as magnitude and phase

When you select this check box, the scope splits the display into a magnitude plot and a phase plot. By default, this check box is cleared. If the input signal has complex values, the scope plots the real and imaginary portions on the same axes. These real and imaginary portions appear as different-colored lines on the same axes, as shown in the following figure.



Selecting this check box and clicking the **Apply** or **OK** button changes the display. The magnitude of the input signal appears on the top axes and its phase, in degrees, appears on the bottom axes. See the following figure.



This feature is useful for complex-valued input signals. If the input is a real-valued signal, selecting this check box returns the absolute value of the signal for the magnitude. The phase is 0 degrees for nonnegative input and 180 degrees for negative input.

Y-limits (Minimum)

Specify the minimum value of the *y*-axis.

When you select the **Plot signal(s) as magnitude and phase** check box, the value of this property always applies to the magnitude plot on the top axes. The phase plot on the bottom axes is always limited to a minimum value of -180 degrees.

Y-limits (Maximum)

Specify the maximum value of the *y*-axis.

When you select the **Plot signal(s) as magnitude and phase** check box, the value of this property always applies to the magnitude plot on the top axes. The phase plot on the bottom axes is always limited to a maximum value of 180 degrees.

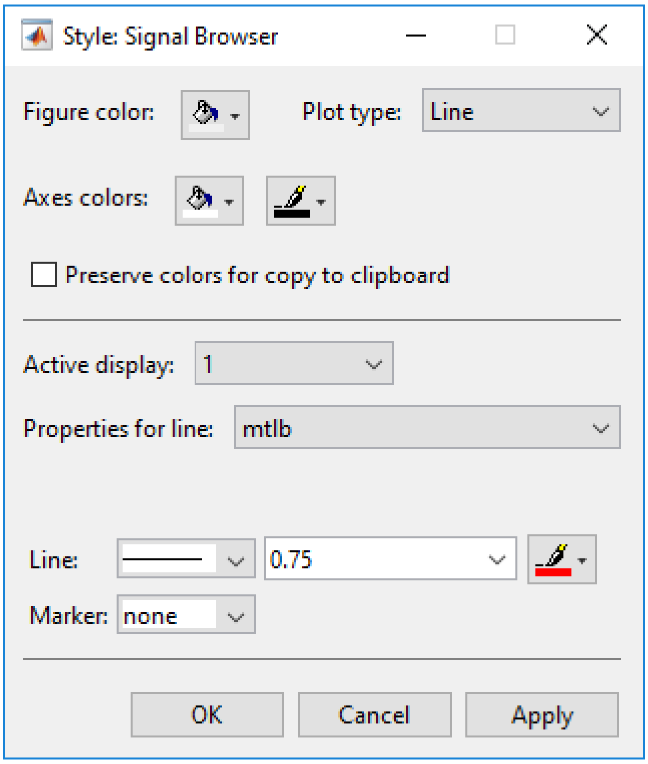
Y-label

Specify the text for the scope to display to the left of the *y*-axis.

This property becomes invisible when you select the **Plot signal(s) as magnitude and phase** check box. When you enable that property, the *y*-axis label always appears as `Magnitude` on the top axes and `Phase` on the bottom axes.

Style Dialog Box

In the **Style** dialog box, you can customize the style of displays. You can change the color of the figure containing the displays, the background and foreground colors of display axes, and properties of lines in a display. From the Signal Browser menu, select **View** > **Style** to open this dialog box.



Properties

The **Style** dialog box allows you to modify the following properties of the Signal Browser:

Figure color

Specify the color that you want to apply to the background of the Signal Browser. By default, the figure color is gray.

Plot type

Specify the type of plot to use. The default setting is `Line`. Valid values for **Plot type** are:

- `Line` — Displays input signal as lines connecting each of the sampled values. This approach is similar to the functionality of the MATLAB `line` or `plot` function.
- `Stairs` — Displays input signal as a *stairstep* graph. A stairstep graph is made up of only horizontal lines and vertical lines. Each horizontal line represents the signal value for a discrete sample period and is connected to two vertical lines. Each vertical line represents a change in values occurring at a sample. This approach is equivalent to the MATLAB `stairs` function. Stairstep graphs are useful for drawing time history graphs of digitally sampled data.

Select display

Specify the active display as a number, where a display number corresponds to the index of the input signal. The number of a display corresponds to its column-wise placement index. The default setting is `1`. Set this parameter to control which display should have its axes colors, line properties, marker properties, and visibility changed.

Axes colors

Specify the color that you want to apply to the background of the axes for the active display.

Preserve colors for copy to clipboard

Specify whether or not to use the displayed color of the scope when copying.

When you select **File** > **Copy to Clipboard**, the software changes the color of the scope to be printer friendly (white background, visible lines). If you want to copy and paste the scope with the colors displayed, select this check box.

Default: Off

Properties for line

Specify the signal for which you want to modify the visibility, line properties, and marker properties.

Visible

Specify whether the selected signal on the active display should be visible. If you clear this check box, the line disappears.

Line

Specify the line style, line width, and line color for the selected signal on the active display.

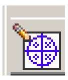
Marker

Specify marks for the selected signal on the active display to show at data points. This parameter is similar to the `Marker` property for the MATLAB Handle Graphics® plot objects. You can choose any of the marker symbols from the following table.

Specifier	Marker Type
none	No marker (default)
○	Circle
□	Square
×	Cross
•	Point
+	Plus sign
✳	Asterisk
◇	Diamond
▽	Downward-pointing triangle
△	Upward-pointing triangle
◁	Left-pointing triangle
▷	Right-pointing triangle
☆	Five-pointed star (pentagram)
✡	Six-pointed star (hexagram)

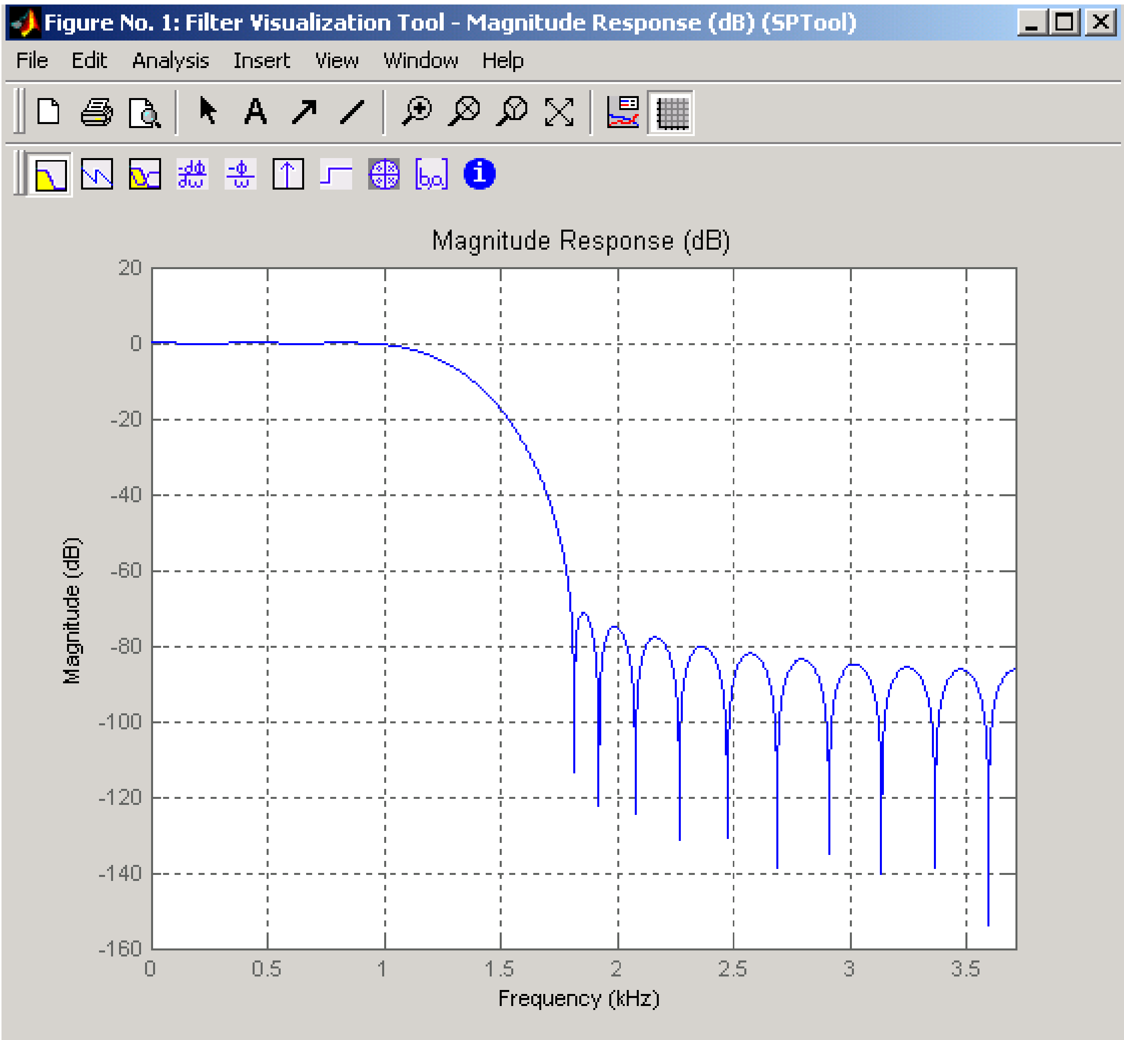
Filter Designer App

The **Filter Designer** app allows you to design and edit FIR and IIR filters. To launch the app, press either the **New** button or the **Edit** button under the **Filters** list box in SPTool.

The Filter Designer app has a Pole/Zero Editor you can access by selecting the  icon in the left column.

Filter Visualization Tool

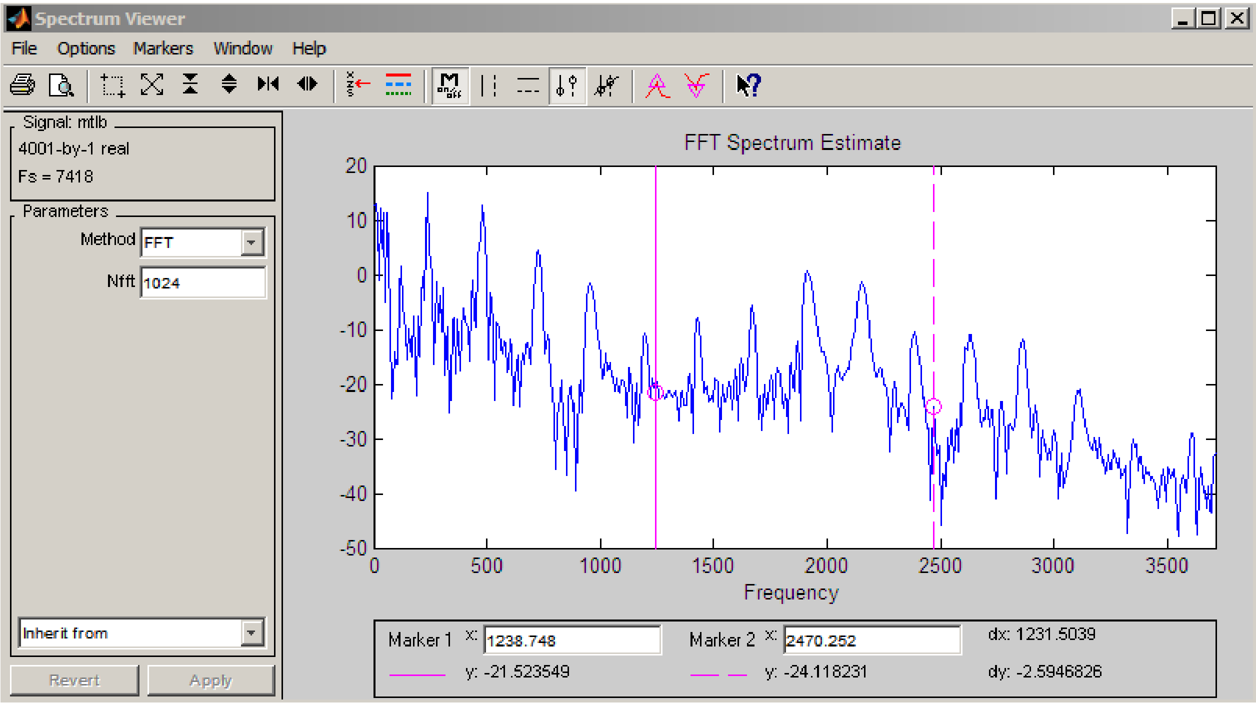
The Filter Visualization Tool (`fvtool`) allows you to view the characteristics of a designed or imported filter, including its magnitude response, phase response, group delay, phase delay, pole-zero plot, impulse response, and step response. To activate FVTool, click the **View** button under the **Filters** list box in SPTool.



Spectrum Viewer

The Spectrum Viewer allows you to analyze frequency-domain data graphically using a variety of methods of spectral density estimation, including the Burg method, the FFT method, the multitaper method, the MUSIC eigenvector method, Welch’s method, and the Yule-Walker autoregressive method. To activate the Spectrum Viewer:

- Click the **Create** button under the **Spectra** list box to compute the power spectral density for a signal selected in the **Signals** list box in SPTool. You may need to click **Apply** to view the spectra.
- Click the **View** button to analyze spectra selected under the **Spectra** list box in SPTool.
- Click the **Update** button under the **Spectra** list box in SPTool to modify a selected power spectral density signal.



In addition, you can right-click in any plot display area to modify signal properties.

Controlling SPTool from the MATLAB Command Line

You can import or export data from SPTool using the command line.

Exporting Component Structures from SPTool

The following commands export component structures from the currently open SPTool:

- `s = sptool('Signals')` returns a structure array of all the signals.
- `f = sptool('Filters')` returns a structure array of all the filters.
- `s = sptool('Spectra')` returns a structure array of all the spectra.
- `[s,ind] = sptool( __ )` returns an index vector indicating which of the elements of `s` are currently selected in SPTool.
- `s = sptool( __ ,0)` returns only the currently selected objects.

Creating and Loading Component Structures

The following commands create component structures and load them into SPTool, opening SPTool if necessary:

- `struc = sptool('create',paramlist)` creates in the workspace a component structure, `struc`, defined by `paramlist`.

  `sptool('load',struc)` loads `struc` into SPTool.
- `struc = sptool('load',paramlist)` loads the component structure defined by `paramlist` into SPTool. If you specify an output argument, then the command also creates a component structure in the workspace.

Example: Create and load a 5th-order Butterworth filter with a cutoff frequency of 0.5π rad/sample. Specify the filter in state-space representation, label it `Butterworth` within SPTool, and set it to filter digital signals sampled at 1 kHz.

```
[z,p,k] = butter(5,0.5);
struc = sptool('create','Filter','zpk',z,p,k,1e3,'Butterworth');
sptool('load',struc)
```

Example: Load into SPTool the periodogram PSD estimate of a 512-sample sinusoidal signal embedded in white noise. Work in normalized units and specify a sinusoid frequency of π/4 rad/sample. Label the spectrum `PSD` within SPTool.

```
n = 0:511;
x = sin(pi/4*n)+randn(size(n))/10;
[pxx,w] = periodogram(x);
sptool('load','Spectrum',pxx,w,'PSD')
```

Example: Create and load a quadratic chirp modulated by a Gaussian. Specify a sample rate of 2 kHz and a signal duration of 2 seconds. Generate a copy of the structure in the workspace.

```
t = 0:1/2000:2-1/2000;
q = chirp(t-2,4,1/2,6,'quadratic',100,'convex').*exp(-4*(t-1).^2);
Chirp = sptool('load',q,2000)
```

```
Chirp =

  struct with fields:

             data: [4000×1 double]
               Fs: 2000
             type: 'vector'
         lineinfo: []
    SPTIdentifier: [1×1 struct]
            label: 'sig'
```

The parameters in `paramlist` must be input in the following order:

Component	**`paramlist` Parameters**
Signals	`component_name`, `data`, `fs`, `label`
Filters	`component_name`, `form`, `filter_params`, `fs`, `label`
Spectra	`component_name`, `data`, `f`, `label`

The parameters are defined as follows:

Parameter	**Definition**
`component_name`	Specify as one of `'Signal'`, `'Filter'`, or `'Spectrum'`. If omitted, `component_name` defaults to `'Signal'`.
`form`	Form or structure of a filter. Specify as one of `'tf'`, `'ss'`, `'sos'`, or `'zpk'`.
`data`	Vector of doubles representing a signal or spectrum.
`filter_params`	Filter representation. • Specify `num` and `den` when `form` is `'tf'`. • Specify an SOS matrix when `form` is `'sos'`. • Specify `z`, `p`, and `k` when `form` is `'zpk'`. • Specify `A`, `B`, `C`, and `D` when `form` is `'ss'`.
`fs`	Optional parameter that specifies the sample rate. If omitted, `fs` defaults to 1.
`f`	Frequency vector. This parameter applies only if `component_name` is `'Spectrum'`.
`label`	Optional parameter that specifies the variable name of the component within SPTool. If omitted, `label` defaults to: • `'sig'` if `component_name` is `'Signal'` • `'filt'` if `component_name` is `'Filter'` • `'spec'` if `component_name` is `'Spectrum'`

See Also

Apps

Filter Designer | Signal Analyzer

Functions

`findpeaks` | `fvtool`

Introduced before R2006a

How useful was this information?