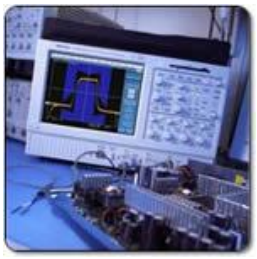
Getting Started Guide

Using MATLAB with Tektronix Oscilloscopes Over a GPIB Connection



It is easy to use MATLAB with your Tektronix oscilloscope over a GPIB connection. The goal of this guide is to allow you to verify within 15 minutes that you can use MATLAB with your Tektronix oscilloscope. MATLAB enables Tektronix oscilloscope users to acquire and analyze data, graphically visualize data, make custom measurements, generate reports, and develop automated applications.

MathWorks wants you to be successful evaluating MATLAB. If you have any questions or problems completing the steps in this guide, please email Brian Madsen at brian.madsen@mathworks.com.

Contact Brian if you would like a Getting Started Guide for using your instrument with a LAN (TCP/IP), Serial (RS-232), or USB connection.

System requirements for completing this guide:

1. A Tektronix oscilloscope with a GPIB interface supporting remote instrument communication.
2. MATLAB and MATLAB Instrument Control Toolbox installed on a computer.
3. VISA software installed on the computer. VISA is often available as a free or low cost download from instrument manufacturers such as Agilent and Tektronix.
4. Appropriate MATLAB Instrument Driver for your instrument. MATLAB instrument drivers are available at: www.mathworks.com/tektronix.

Email Brian at brian.madsen@mathworks.com if you would like a trial of MATLAB and the Instrument Control Toolbox, need assistance finding a MATLAB Instrument Driver for your instrument, or if you have any questions on the system requirements.

Prepare you system

Step 1: Start Test and Measurement Tool

Start MATLAB and issue the following command to launch Test and Measurement Tool:

```
tmtool
```

Test and Measurement Tool (Figure 1) is the graphical user interface for configuring and controlling instruments in MATLAB.

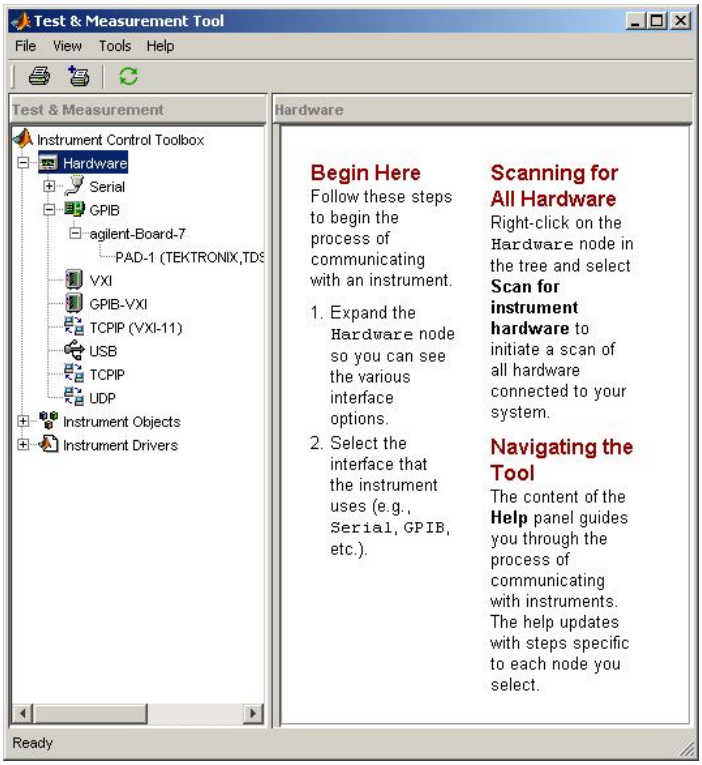


Figure 1: The Test and Measurement Tool from the Instrument Control Toolbox

Step 2: Connect your instrument

Connect your instrument to your computer via a GPIB interface. Supported interfaces are listed at http://www.mathworks.com/products/instrument/supportedio7141.html.

On your Tektronix oscilloscope, select **Utilities** > **GPIB Configuration**, and ensure that under the **GPIB Configuration** panel the **Talk/Listen** button is selected.

In Test and Measurement Tool, right click on the Hardware node and choose **Scan for instrument hardware**. Your instrument will be listed under the `GPIB` node. As you can see from Figure 1, there is a Tektronix scope (primary address 1) connected via an Agilent GPIB adapter (board index 7).

Step 3: Locate an instrument driver

MATLAB with the Instrument Control Toolbox supports instrument drivers, which allow you to communicate with your device without knowing low-level instrument commands. To see if your instrument driver is already installed, right click on the `Instrument Drivers` node and select **Scan for Instrument Drivers**. Under the `Instrument Drivers` node, look for a driver that matches your device. If you do not have a matching driver installed, more are available from: http://www.mathworks.com/matlabcentral/reports/fileexchange/instrument.html. If you are unable to find a matching driver, a driver for a similar device may work with little or no modification.

Downloaded drivers must be placed in a directory on the MATLAB path such as **<matlabroot>\work**. Right-click on `Instrument Drivers` and choose **Scan for Instrument Drivers** to refresh the current list of drivers.

Connect to the instrument

Step 4: Create the interface object

Navigate to the `Interface Object` node under the `Instrument Objects` node. Click on the **New Object** button in the lower right. The resulting dialog is shown in Figure 2.

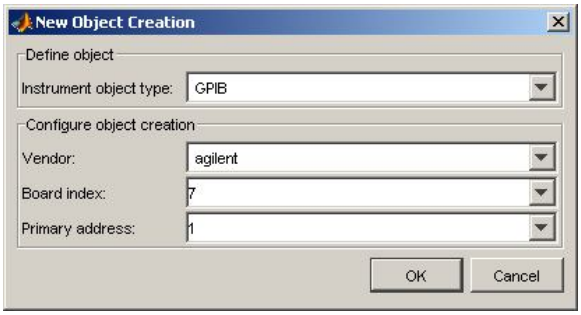


Figure 2: Interface creation dialog

Configure this to match your GPIB connection (from Step 2) to the instrument and click **OK**.

Step 5: Create the device object

Under the `Instrument Drivers` node, right click on the instrument driver for your device and select **Create Device Object Using Driver** as in Figure 3. In this example, the instrument is a Tektronix TDS5104B. In the resulting dialog box, select the interface object you created in Step 4.

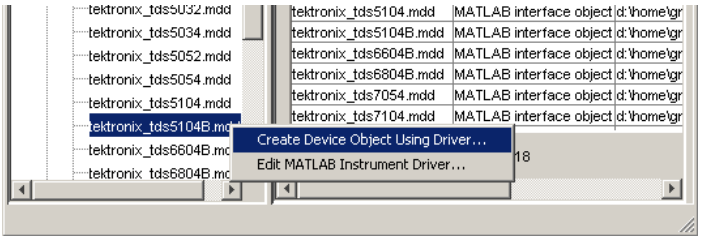


Figure 3: Creating an Instrument Object from a driver

Click **OK** to complete the creation of the device object.

Step 6: Connect to the device object

Select the newly created instrument object, which in this example is labeled scope-tektronix_tds5104B, under the `Device Objects` node. Now click **Connect** in the upper right hand corner and you are ready to start interacting with your device.

Interact with the instrument

Step 7: Query a property

Select the instrument object, click on the **Properties** tab, navigate to the `Timebase` property in the `Acquisition` group, and click **Get** as in Figure 4. The results appear in the **Response** region in the right side.

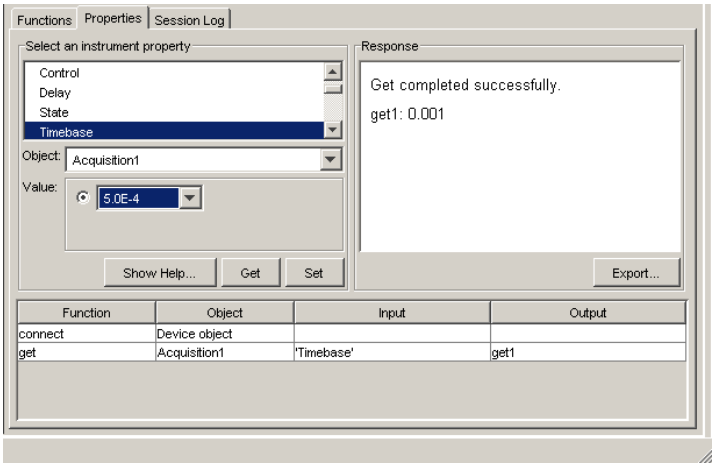


Figure 4: The result of getting the `Timebase` property

Step 8: Configure a property

Navigate to the `Timebase` property then choose a value for the property and click **Set**. The **Response** region indicates if the set succeeded or displays an error message indicating the reason if it failed.

Step 9: Invoke a function

View the instrument's functions by choosing the **Functions** tab. To call the `readwaveform` function, first scroll to `readwaveform` under the `Waveform` group in the function list. Then enter input argument **`'channel1'`** (include the single quotes) to read the first channel. Next enter output arguments *Y, X, YUNIT, XUNIT, HEADER* as a comma separated list. (Note: The HEADER argument may not be needed.) Finally, click **Execute** to read the waveform. The results appear in Figure 5.

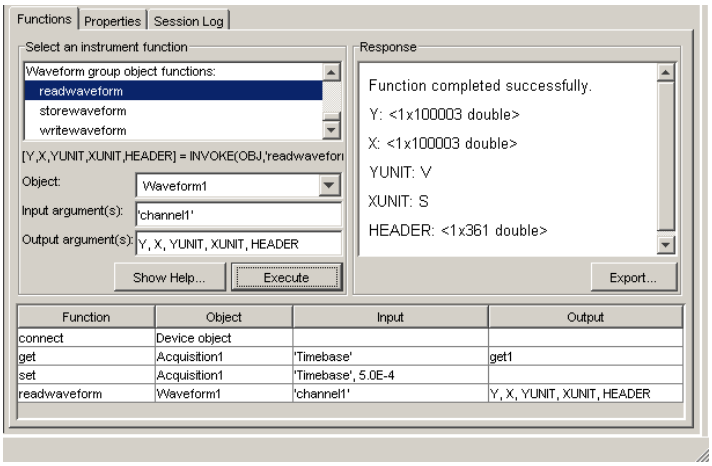


Figure 5: Invoking `readwaveform` of an instrument object

Step 10: Visualize results

Choose the menu item **File > Export > Instrument Responses**. To plot the waveform (variable *Y*) from the resulting dialog (Figure 6), check only the box to the left of variable name Y and uncheck all others.

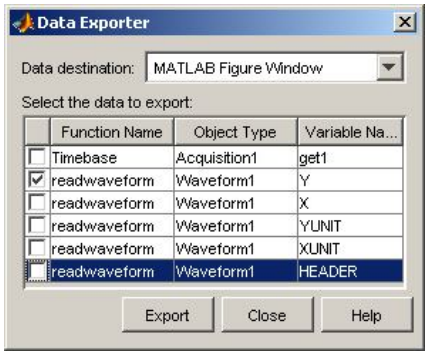


Figure 6: Data export dialog

In the **Data Destination** drop-down, choose **MATLAB Figure Window** and click **Export**. This will create a MATLAB Figure window displaying the waveform you captured.

Step 11: Analyze results

You may further analyze your data by exporting it to the MATLAB workspace. From the **Data Exporter** (Figure 6), choose **MATLAB Workspace** as the data destination then click **Export**. Note: Once your values and waveforms are in the MATLAB workspace, you may use them with MATLAB functions or scripts including ones you create (not shown here).

Step 12: Reuse your session

Test and Measurement Tool automatically generates MATLAB code for all of your interactions with your instrument. Select the **Session Log** tab of your instrument object to view the MATLAB code (Figure 7). Furthermore, using the **Save Session** button, you may export this code as an M-file which you can use to repeat your steps or develop a larger application.

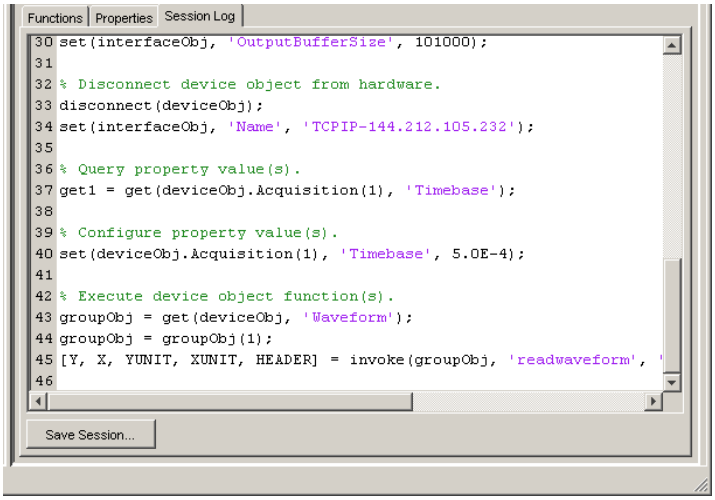


Figure 7: Viewing the code from your session

If you have successfully completed all these steps, you have verified that you can use MATLAB and the Instrument Control Toolbox with your instrument.