Real-Time Workshop in MATLAB

Jitkomut Songsiri

July 5, 2011

This note provides a short guideline for setting up a MATLAB simulink file to acquire data from a hardware. To be able to use this feature, you must have:

1. a data acquisition board and its driver installed properly in your PC,

2. included **Real-Time Workshop** toolbox while installing MATLAB (or you may add it later).

In this document, we have tested with NI (National Instruments) PCI-6024E card with 16 analog inputs at up to 200 kS/s, 16-bit resolution and up to 2 analog output at 10 kS/s.

1 Creating a model

To create a Simulink file, follow these steps:

1. Open the Simulink library browser and browse for **Real-Time Window Target**
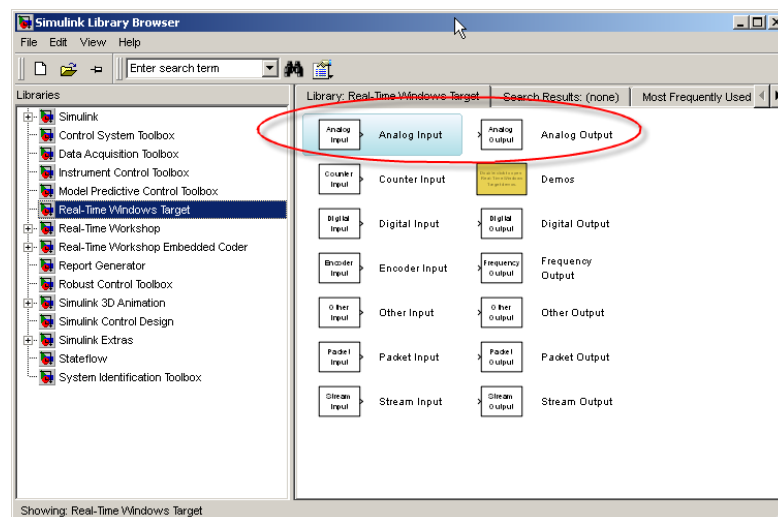

2. The two important blocks are **Analog Output** and **Analog Input**. Insert these two in your Simulink file.

3. Analog input is used for receiving data from an external source. Double click at this block, you will see the dialog box as shown in the follwing figure. Click at

Install new board and select the device you use. MATLAB should recognize a wide range of boards from various companies. In box **Sample Time**, we put a variable `Ts` (and will assign a value to it later in the Command window.) In box **Input Channels**, put a vector containing the numbers of channels you want to acquire data from. Here we use channel 1,2,3, and 4. Other parameters (such as input range) should be automatically recognized from MATLAB after we added the board.
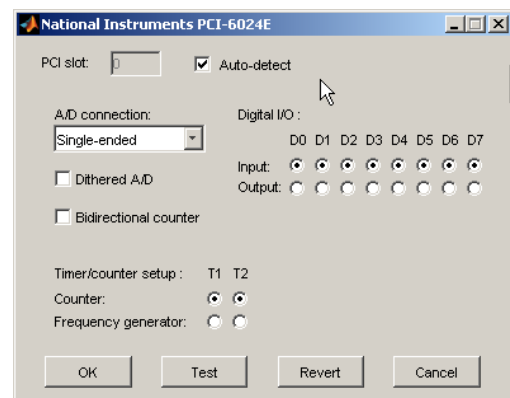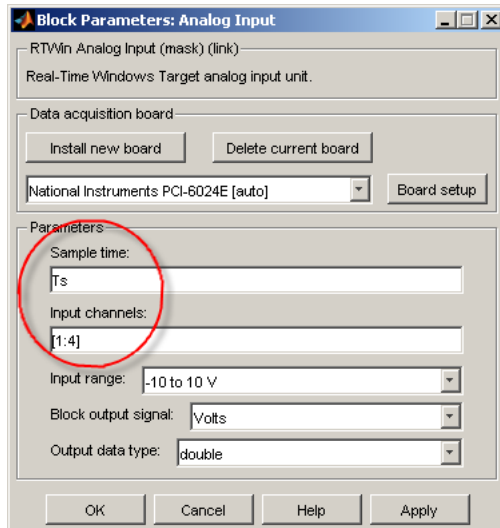

If you click at the **Board Setup**, it shows the configuration of the board. We basically change nothing, since the default setting works fine.

4. Analog output is used for generating a signal from MATLAB and sending it to an external sink. Double click at this block, the parameters are similar to the analog input block. Select your card, define the sample time (use `Ts`) and select the output channel. Here we use the first channel, so we put 1.
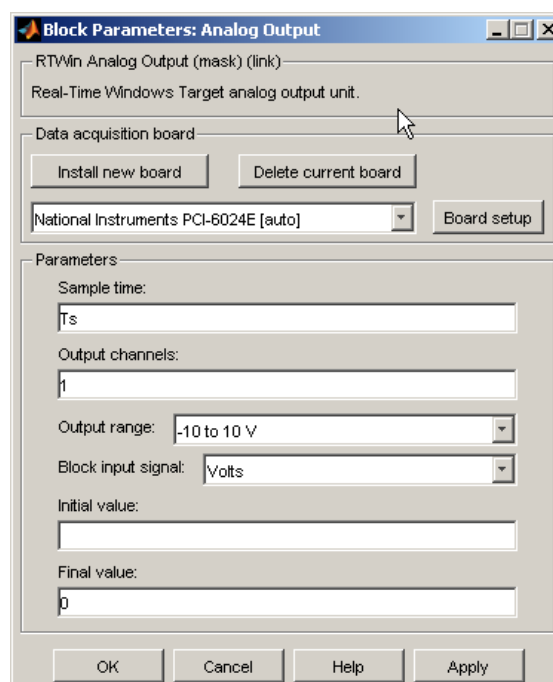
5. You may build a simple scheme to test how it works by using a scope as a sink to Analog input block, and use a signal generator as a source to Analog output.
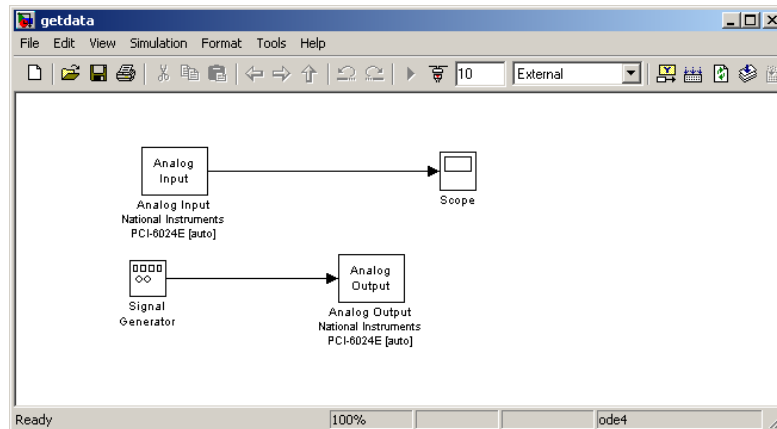

6. For example, we use a sine signal with amplitude 1 and frequency 0.2 Hz (make sure it is not faster than the sampling rate of your card). In scope block, click at **Parameters** and set parameters in **Data History** tab. Uncheck the box **Limit data points to last** and check the box **Save data to workspace**. We use variable `ys` in array format (for some reason, we got an error while running real-time with `ys` in structure format). With this setting, after we run the simulation, the variable `ys` will appear in the current workspace and has 5 columns; the first column is the time vector and the remaining 4 columns are the data from the 4 channels.
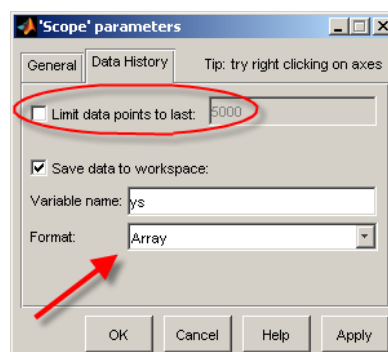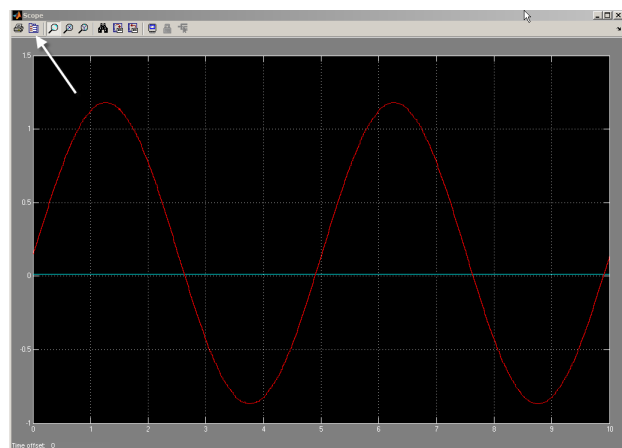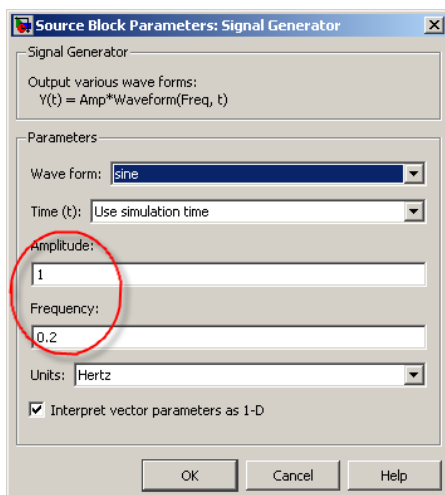
2 Setting Simulation Parameters

On your Simulink window, click at **Simulation/Configuration Parameters** and set these parameters:

1. Select **Solver** from the left menu. Change **Solver options** to **Fixed-Step** type and specify a fixed-step size. Here we put a variable `Ts` (same as the sample time.) This may not be necessary, but the step size should not be less than the sample time.
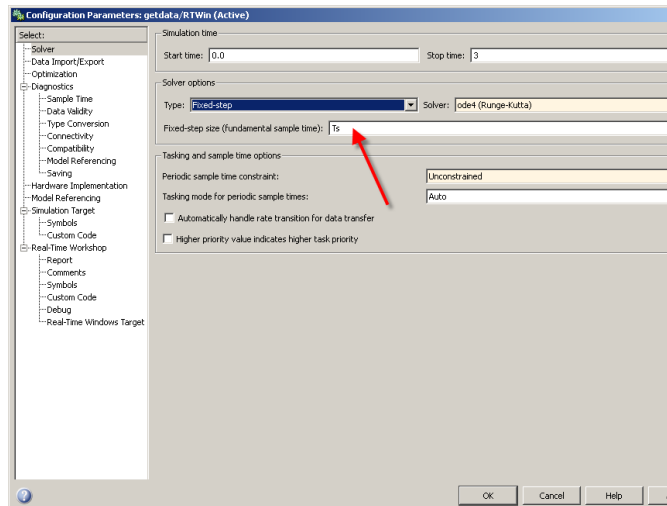
2. Select **Data Import/Export** from the left menu. We uncheck the boxes in **Save to workspace** since we already set data logging in the scope. If you use a workspace to get the data instead of using a scope, you may want to uncheck the box **Save option/Limit data points to last**.
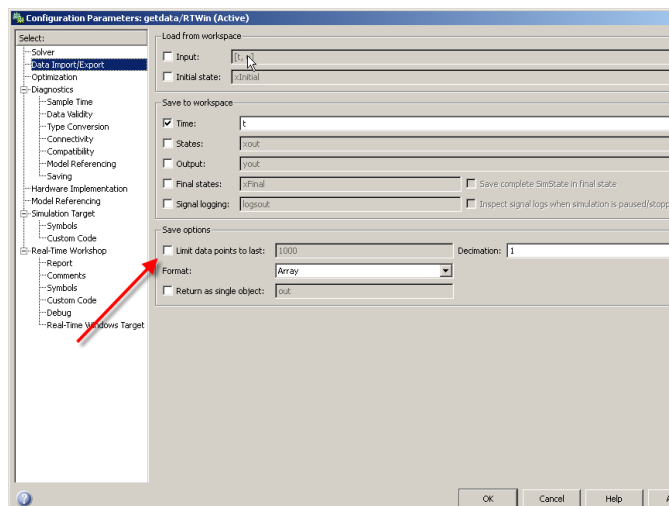
3. Go to **Real-Time Workshop** menu on the left. At **Target Selection**, click at **Browse** to choose **System target file**. Another dialog box will appear. Choose **Real-Time Windows Target**.

4
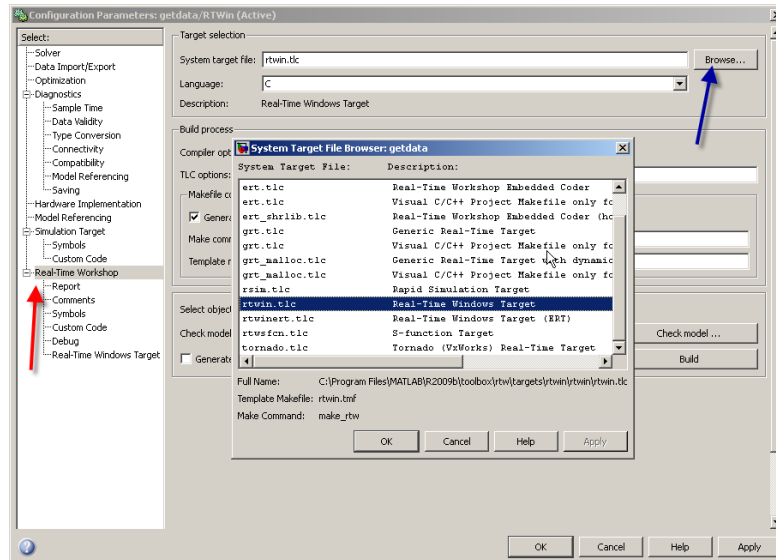
4. After setting these parameters properly. Close the dialog box of Configuration Parameters.

5. On your simulink window, change from **normal** to **external** mode. In external mode, you are not able to use some keyboard shortcuts such as oopy (Ctrl-C) or paste (Ctrl-V) as in the normal mode.
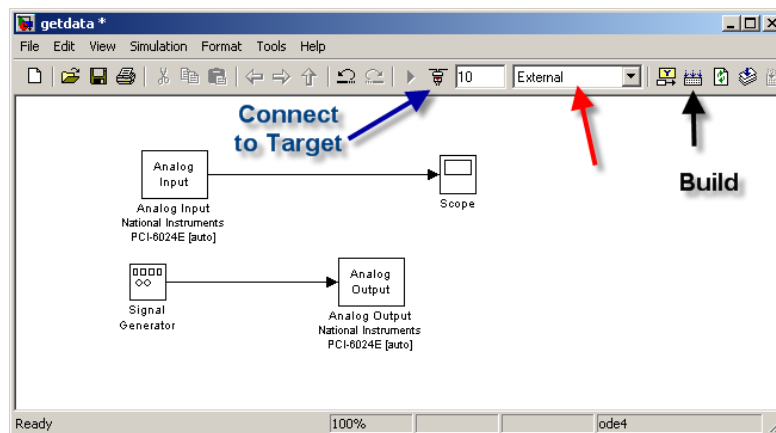

3 Data logging (optional)

The number of data points from the simulation obviously depends on the simulation time and the step size we use. We often prefer not to limit the data points to last at a certain number. Apart from the settings described previously, this parameter is also dependent of the setting in the menu **Tools/External Mode Control Panel** in the simulink window. This is for running a simulation in real-time only.

1. Click at **Signal & Triggering** button.
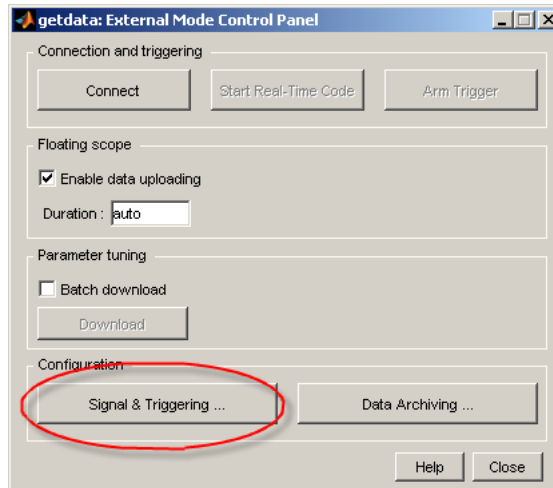
2. The dialog box of **External Signal & Triggering** will appear. In the **Duration** box, enter the number of sample points in a data buffer.
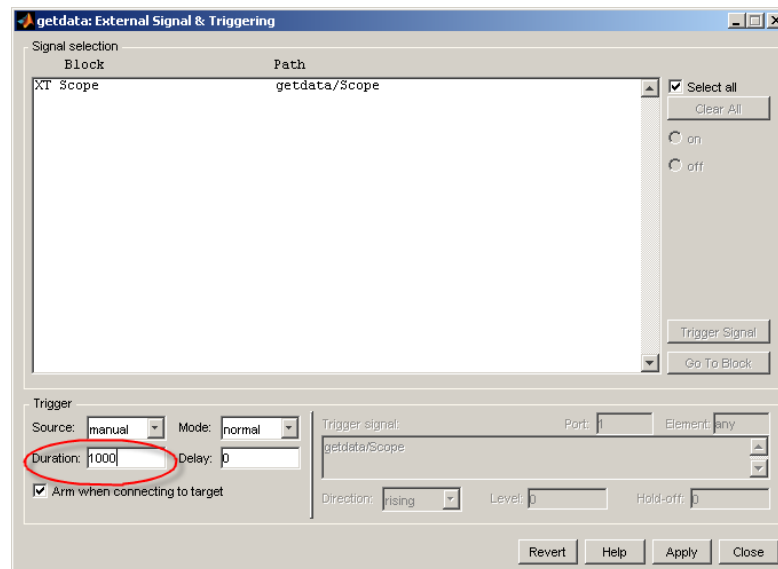

The **Duration** value is related to the **Limit data points to last** value in the **scope parameters** box. The smaller of either value limits the number of sample points saved to the MATLAB workspace.

4   Running the simulation

Suppose we have connected the hardware properly. Since we have set the parameter sample time as `Ts`, we assign a value 0.01 to `Ts` in the command window.

1. Before running a simulation, the simulink file must be **built** by pressing the button shown in the figure (black arrow). Some codes will be executed in the Command window.
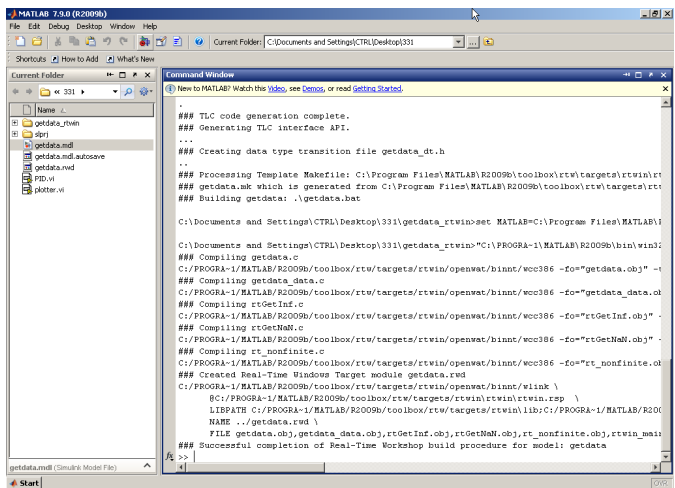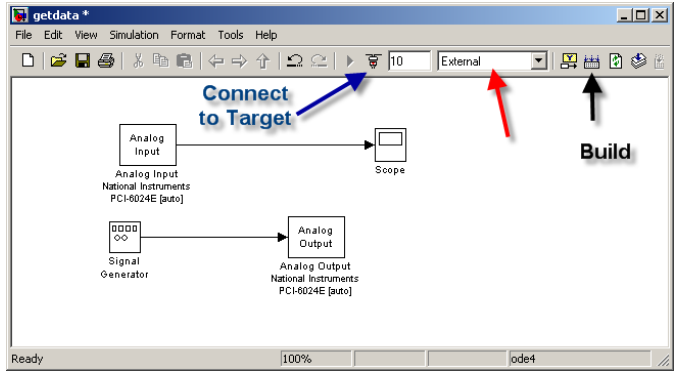
2. If there is no error on the Command window, next, **Connect to a target** by clicking at the button next to the blue arrow. Now you are ready to run the simulation.

5   Input Channels

This is for my own reference only. We use 4 input channels from NI-PCI 6024E card and the 4 probes have the description as follows.

Channel	Probe color	Description	Graph color	
			workspace	scope
1	green	reference signal	blue	yellow
2	yellow	output signal	green	magenta
3	brown	control signal	red	light blue
4	white	error signal	light blue	red

References

[1] Real-Time Windows Target User's guide: For Use with Real-Time Workshop, The Mathworks, version 2, page 3-33.