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:

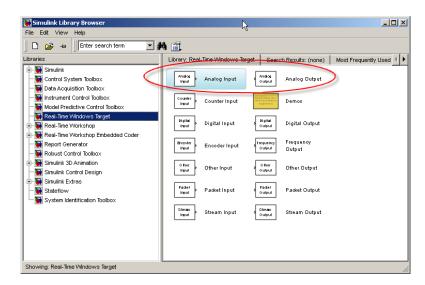
- 1. a data acquisition board and its driver installed properly in your PC,
- 2. included **Real-Time Workshop** toolbox while installing MATLAB (or you may add it later).

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 following 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.

👍 Block Parameters: Analog Input
RTWin Analog Input (mask) (link)
Real-Time Windows Target analog input unit.
Data acquisition board
Install new board Delete current board
National Instruments PCI-6024E [auto] Board setup
Sample time: Ts Input channels: [1:4]
Input range: -10 to 10 V
Block output signal: Volts
Output data type: double
OK Cancel Help Apply

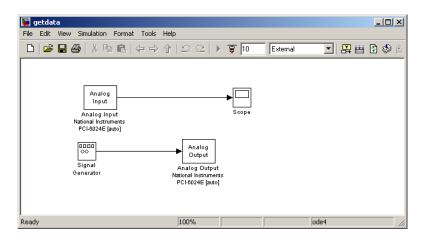
📣 National Instruments PC	II-6024E	
PCI slot: 🛛	Auto-detect	I
	\searrow	
A/D connection:	Digital I/O :	
Single-ended	D0 D1 D2 D3 D4 D	05 D6 D7
_	Input: •••••	
Dithered A/D		000
Bidirectional counter		
Timer/counter setup : T	1 T2	
eesinten.		
requere, generator.		
OK Tes	t Revert C	Cancel
		ancor

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

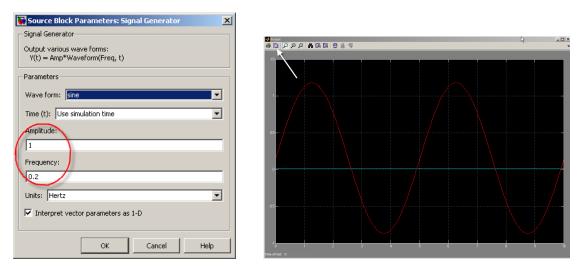
4. Analog output is used for generating a signal from MATLAB and sending it to an external sink. Double click at this block, the parameters are similar to the analog input block. Select your card, define the sample time (use Ts) and select the output channel. Here we use the first channel, so we put 1.

🥠 Block Parameters: Analog Output
RTWin Analog Output (mask) (link)
Real-Time Windows Target analog output unit.
Data acquisition board
Install new board Delete current board
National Instruments PCI-6024E [auto]
Parameters
Sample time:
Ts
Output channels:
1
Output range: -10 to 10 V
Block input signal: Votts
Initial value:
Final value:
p
OK Cancel Help Apply

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.

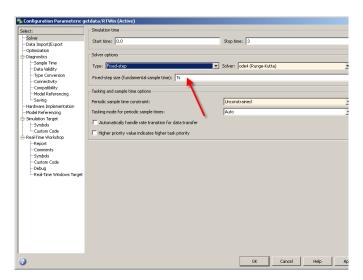


📣 'Scope' pa	arameters			<u> </u>
General Da	ta History	Tip: try righ	t clicking	on axes
Limit data	a points to last	5000		
🔽 Save dat	a to workspac	e:		
Variable nar	ne: ys			
Format:	Array			-
	1			
	OK Ca	ncel He	alp	Apply

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.



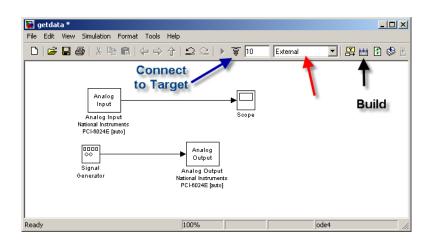
2. Select **Data Import/Export** from the left menu. We uncheck the boxes in **Save to workspace** since we already set data logging in the scope. If you use a workspace to get the data instead of using a scope, you may want to uncheck the box **Save option/Limit data points to last**.

Select:	Load from workspac	е					
Solver Data Import/Export	🗆 Input: 🛛 🗍	^{t,} []					
Optimization	🗌 Initial state: 🛛	<initial< td=""><td></td><td></td><td></td><td></td><td></td></initial<>					
-Diagnostics							
Sample Time Data Validity	Save to workspace						
- Type Conversion	Time:	t					
Connectivity	States:	-					
Compatibility		xout					
Model Referencing	Cutput:	yout					
Saving Hardware Implementation	Final states:	×Final		🗖 🇖 Save d	omplete SimState	e in final state	
Model Referencing	🗌 Signal logging:	logsout		🗌 🗖 Inspec	t signal logs whe	n simulation is pause	d/stoppe
Symbols	Save options						
E-Real-Time Workshop	Limit data points	to last:	1000	Decimation:	1		
Report	Format:		Array				
Comments							
Custom Code	Return as single	object:	out				
Debug							
Real-Time Windows Target							
· ·							
0				ОК	Cancel	Help	Ap

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**.

Select:	- Target selection				
Solver Data Import/Export	System target file:	rtwin.tlc			Browse
Optimization	Language:	c			
Diagnostics	Description:	Real-Time Windows Target			_
Sample Time Data Validity	bosciption	ited fille filleono farget			
··· Type Conversion	Build process				
Connectivity	Compiler opt	ystem Target File Brows	er: getdata	X	1
Compatibility	970	tem Target File:	Description:		
Model Referencing	TLC options:	tle	Real-Time Workshop	Enbedded Coder	
·Saving Hardware Implementation	Makefile cc	.tlc		ect Makefile only fc	
Model Referencing	General ert	_shrlib.tlc		Embedded Coder (hc	
Simulation Target	Make comr gr1	tle	Generic Real-Time	Target	
Symbols	gri	tlc		ect Makefile only fo	
Custom Code		_malloc.tlc		Target with dynamic	
Real-Time Workshop		_nalloc.tlc		ect Makefile only fo	
Comments		in.tlc	Rapid Simulation 1 Real-Time Windows		
Symbols		vinert.tlc	Real-Time Windows		
Custom Code		sfcn.tlc	S-function Target	Target (akt)	Check model
Debug	to	mado.tlc	Tornado (VxWorks)	Real-Time Target	Chockmood III
¹ Real-Time Windows Target	🗌 🗌 Generata 🔒				Build
·	Full r	lame: C:\Program F	iles\MATLAB\R2009b\toolbox\r	rtw(targets)rtwin)rtwin/rtwin.tlc	
	Tem	late Makefile: rtwin.tmf			
	Make	Command: make rtw			
			OK Cancel	Help Apply	

- 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.

🕽 getdata: External Mode Control Panel 📃 🔍 🗙
Connection and triggering
Connect Start Real-Time Code Arm Trigger
Floating scope
✓ Enable data uploading
Duration : auto
Parameter tuning
E Batch download
Download
Configuration
Signal & Triggering Data Archiving
Help Close

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

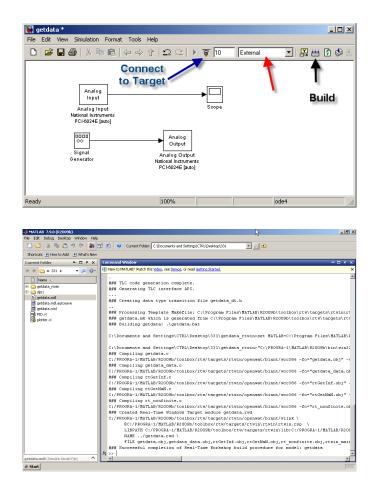
📣 getdata: External Signal & Triggerin	g	
Signal selection		
Block	Path	
XT Scope	getdata/Scope	Select all
		Clear All
		C on
		C off
		0.01
		Trigger Signal
		Go To Block
-		
Trigger		
Source: manual - Mode: normal	Trigger signal: Port: 1	Element: any
Duration: 1000 Delay: 0	getdata/Scope	A
buration. pool		*
Arm when connecting to target	Direction: rising - Level: 0	Hold-off: 0
	Breaten Insing	now on p
	RevertH	elp Apply Close

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
	1 1006 00101	Description	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.