CMPE-432. Feedback Control Systems. Lab #2.*
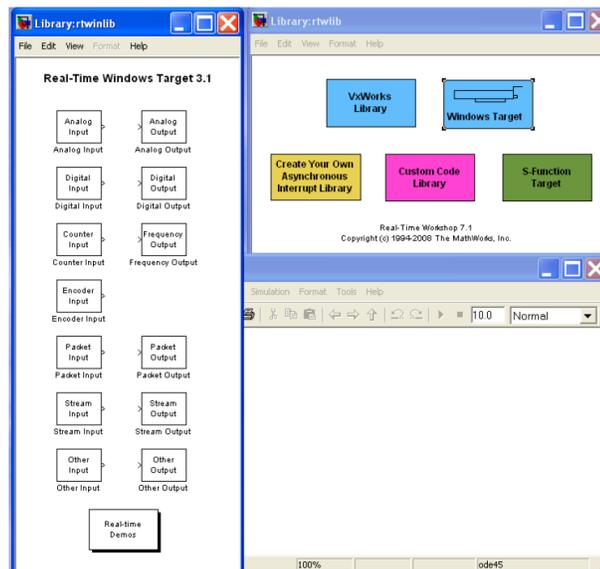
Spring 2010.

Date March 10th, 2010.

Real Time Workshop

Steps involved :

1. Write "rtwlib" on Command Window of MATLAB.

2. Select the Real Time Windows Target library.

3. Select the desired input/output blocks from the library.

4. Drag the selected blocks onto your Simulink model.

---

5. Double click the I/O block to bring up its properties box. Install the desired board from the list. Control System Lab at LUMS provides DAQ card of National Instruments. So select NI PCI 6221 from the list.

6. Select the sample time of your desire.

7. Select the ports you are trying to use from board, Check them if you want to use them as output port and leave unchecked if you want to use them as an input port.

8. Set final and start value from its specified field, strictly set FINAL VALUE as 0. So that when model finishes its run, it forces the output voltage to the DC motor to be 0V.
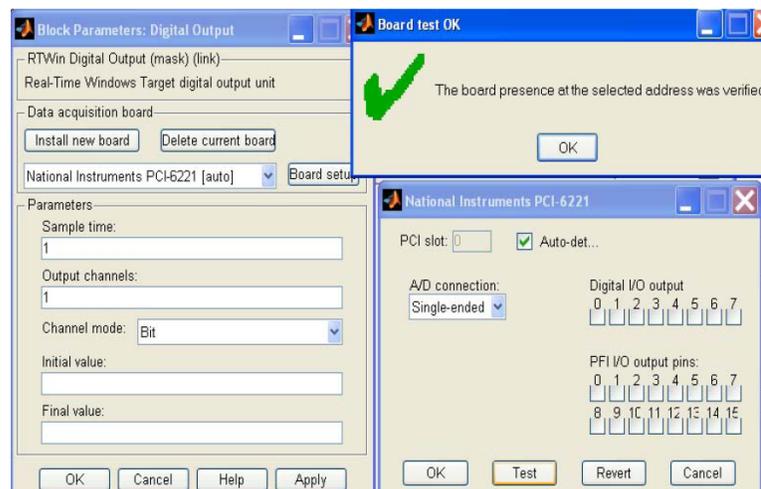

9. Now set the configuration parameters inside SIMULATION pull down menu.

10. As Real time Simulink model can run only with a Fixed Step Size. So on the left side panel select SOLVER, and in SOLVER OPTIONS area, select FIXED STEP for solver type. Choose an appropriate SOLVER for your model. If all blocks are discrete time blocks then discrete solver is a good option. Otherwise you can select Ode4 solver mentioned there.

11. Set your fixed step size same as mentioned in the sampling time field of the I/O block properties.

12. Next, select REAL-TIME WORKSHOP on the left.

13. In the TARGET SELECTION area, click BROWSE, and scroll down to the bottom, and select rtwin.tlc from the menu.
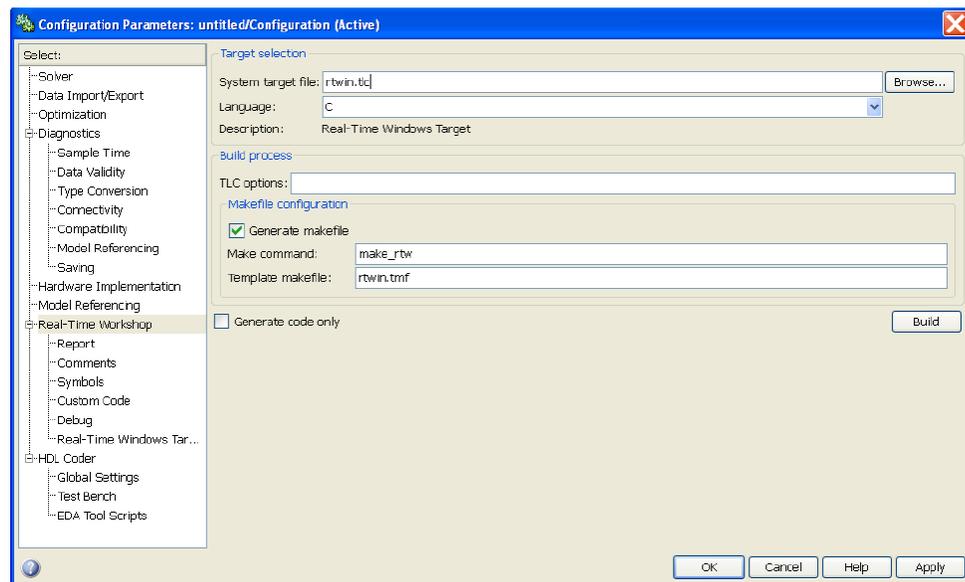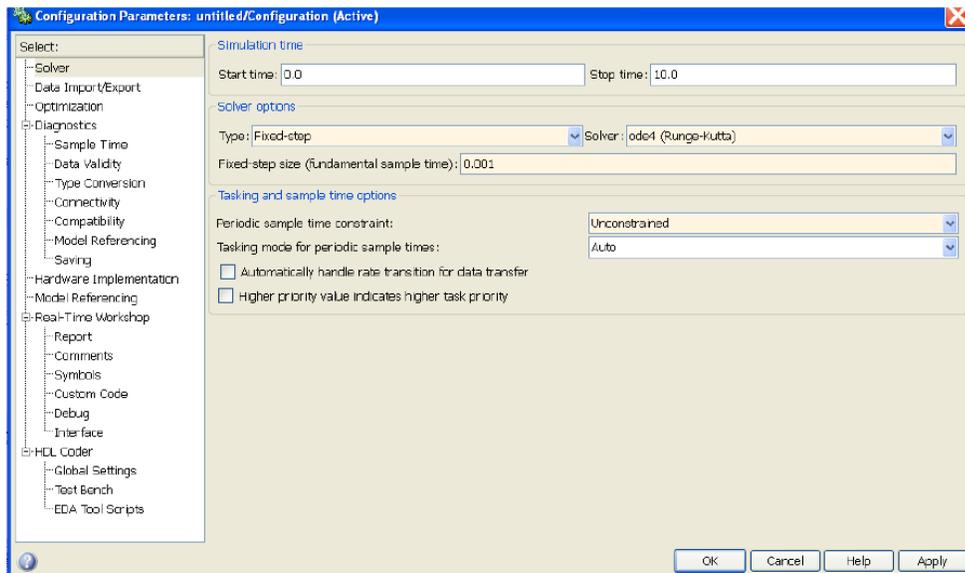
14. Lastly , the model must be set to EXTERNAL mode. In the mode toolbar to the right of the final simulation time there is a pull down menu that usually says NORMAL, click the menu and select EXTERNAL.
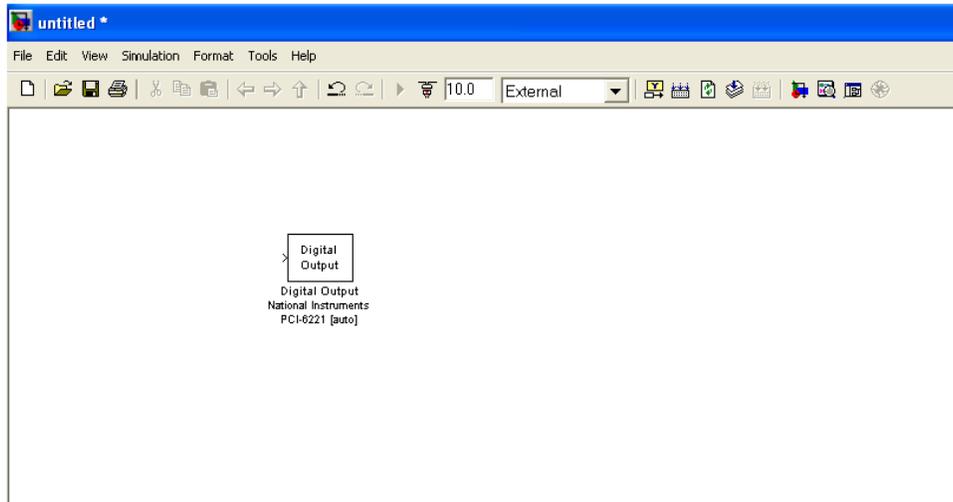
15. Like an oscilloscope, to start the data logging procedure a trigger is needed to initiate the process. A simple approach is to define a dedicated scope block, with one trigger signal entering it, e.g a step function set with a step time similar to sampling time already set. The configuration is shown in figure below.
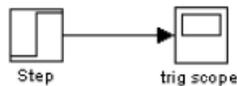
16. Name that scope as TRIGGER SCOPE, then open EXTERNAL MODE CONTROL PANEL in pull down menu of TOOLS.

17. Click SIGNAL AND TRIGGERING button.

18. Select TRIGGER SCOPE block from the list and click TRIGGER SIGNAL button. This selects the input to scope as trigger signal.

19. Then set SOURCE = SIGNAL, MODE = NORMAL, DELAY = 0, DURATION = final time/ sampling time and LEVEL = 0.5. When it crosses 0.5 value it will initiate signal logging.
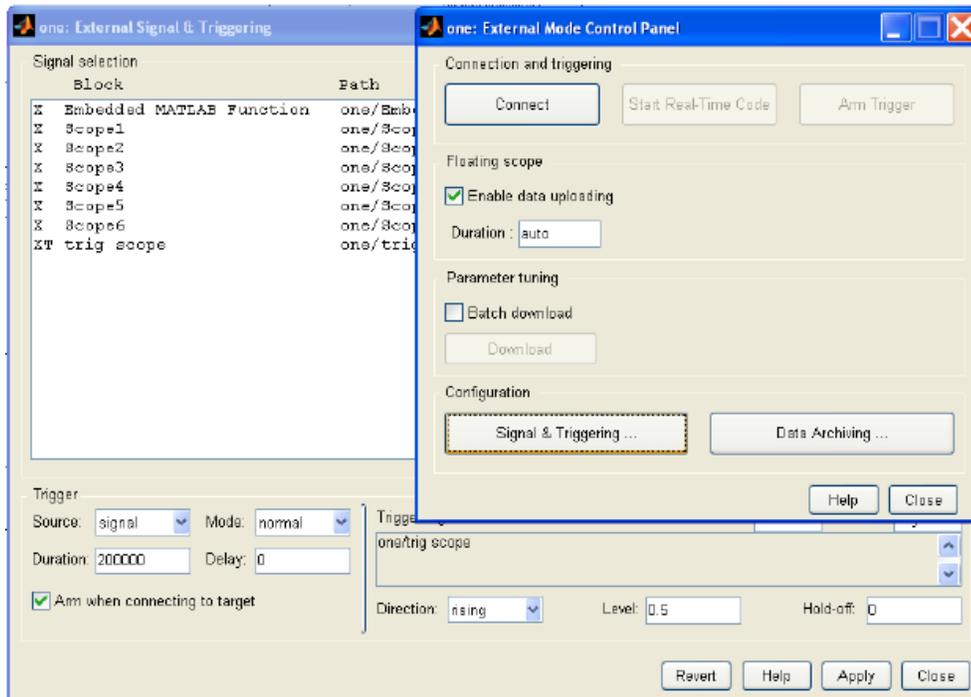
20. The model must be BUILT first before it can be run. Buuilding involves generating and compiling the real time code. To do this click on the TOOLS menu and under Real -Time Workshop select BUILD MODEL, or just hit CTRL+B.

21. Now click CONNECT TO TARGET and then PLAY button will run the model.

Simulink Model of DC motor speed control