dSPACE Tutorial

Building a Simulink file and connecting it to ControlDesk

1. Create your own directory on the PC
2. Download the Simulink file Motor_IO from Social and save it under a different name. Start Matlab 2014B (32 bit).
3. If you want to see how the PWM block works just open it. In the Enc I/F subsystem the counters from the encoders are read and scaled to radians. The encoder I/O block has a second output which is the number of pulses since the last reading, this is used to calculate the velocity. To get the correct scaling for the velocity we need to set the sampling period of the model in each scaling block since the velocity is derived by

$$vel(n) = \frac{pos(n) - pos(n-1)}{T_s}$$

   The best is to enter *Ts* at the Matlab prompt and not as an numeric value in the gain block. The step size of the fixed integration solver, see below must have the same time *Ts* otherwise is the velocity wrongly calculated.
4. Select, Simulation -> Model Configuration Parameters from the menu in the Simulink file. A few changes should be done to prepare Simulink for real-time code generation. These are already set in the model you saved but if you create a new model the following settings must be done.

Some Simulink default parameters are unsuitable for use with RTI.
It is recommended to adapt the Configuration Preferences.

- Solver -	current		recommended setting
Stop time:	"	->	'Inf'
Solver type:	"	->	´Fixed-step´
Fixed-step-size:	"	->	´must be the sampling period *Ts*
-Optimization-			
Block reduction optimization:	"	->	'off'
Signal storage reuse:	"	->	'off’
- Real-Time Workshop -			
RTW system target file:	"	->	'rti1104.tlc'
-Real-Time Workshop			
RTI load options		->	uncheck, Load application after build

5. **IMPORTANT:** change to the correct directory in Matlab at the Matlab prompt, i.e. the same directory that you have saved the Simulink file in. Many files will be created in the **current** Matlab directory.
6. Run, Code -> C C++ Code -> Build Model (Ctrl+B). Code is generated, compiled and linked to a executable. Check that the build procedure has been performed correctly at the Matlab prompt. It should end with the message,
   ### Successful completion.....
7. Start the program ControlDesk 5.3, icon on the desktop.
8. Create a new Project and experiment, a project is a container for all necessary files. Select *File -> New Project + Experiment* from the menu. Create a new *Root directory* and give the project a name, click next and give the experiment a name, click next. Select *DS1104* and click next. Import the *motor_io.sdf* file, it was created during the build process of the Simulink model and should be located in the same directory as the Simulink model.
   Tip! One experiment is always connected to one Simulink file. You can define more than one experiment in a project, e.g., one for an identification model and one or more for control models.
9. The main area is the **Layout window** it is used to build up instrumentation to interact with the real-time process. With input instruments is it possible to change Simulink parameters on the fly and with displays and plotters to watch real-time data from the Simulink diagram. i) Open the *Instrument Selector* window, normally on the right margin, if you can't find it enter the *View* menu -> *Controlbars* and check the *Instrument selector*. ii) From the *Standard Instruments* put One *RadioButton*, one *Display*, one *CheckButton* and one *OnOffButton* on the *Layout* window.
10. Open the Variables window and browse through the hierarchy. Here you will find all the Simulink Block names under their respective hierarchy. Example, the Signal Generator is directly under the Model Root, whereas the grouped blocks PWM and Enc I/F are expandable by clicking the + sign. The signals and parameter of the Simulink blocks are connected to the instruments by simply drag and drop. Each basic block in Simulink has at least one parameter and one output signal.
11. Connect variables to instruments:
    i.) The Display to the position output of the encoder, found on the top level of the EncI/F block.
    ii.) The Radio Button to the signal selector. Configure the Radio Button (properties window), set the value of one of the buttons to 0, and the other to 1. 1 gives a Sine output and 0 a Square output, write an appropriate text for each button.
    iii.) The OnOffButton is used to reset the encoders. Connect it to the Simulink block Reset Encoder which is inside the Enc I/F node. Configure the Button so that its value is 1 when pushed and 0 when it is off and remove the second button, write an appropriate text for the button.
    iiii.) The CheckButton is used to enable the H-bridge, connect it to the parameter in the block enable found in the PWM group.
12. Another way to connect variables and parameters to instruments is by simply drag and drop from the Variable browser or from the checked variables. Select the SineWave block in the Variable browser, check the boxes for both parameters, Amplitude and Frequency. Select the SquareWave block and do the same. Take one of the parameters from the Checked Variables window (to the right) and drag out to the layout, in the new list that pops up select the Variable Array instrument.
13. Start the real-time connection between instruments and the real-time application. Click the *go online* icon, or select from menu.

14. Turn on the DC power supply. **OBSERVE.! Connect the motor boards plus cable to 24V and motor ground to GROUND on the power supply.**
15. Turn the motor by hand and check if the encoders work correctly, e.g., turn the load one revolution and check if the displays show the correct values. Generate a signal from the signal generator and check if the motor starts. Reset the encoders
16. Save the project and close ControlDesk, open it again, open your Project/Experiment, go online and check that it still works.
17. The real-time execution can be stopped or restarted from the Platform/Device window, right click the ds1104 icon and select *Reload application*. This can be useful if a program reset is needed for example if the program has become unstable.
18. When the model structure is changed must new c-code be generated by rebuilding the model. To load the new model code you have to right click the .sdf file in the project manager and select reload variable description.

Plotting signals in ControlDesk

1. Select the variables you want to plot in the variable browser, take the velocity and drag it to the layout from checked variables or drag directly from the variable browser. Select Plotter as instrument.
2. Click the Start Measuring icon
3. **Configure the plotter:**
   Open the *Measurement Configuration* window, expand Acquisition-> Platform-> HostService and select Duration Trigger 1. Explore the functionality of Duration time, Auto Repeat, Stop and Duration. To get triggered plots, select the *Platform Trigger 1* drag the variable that you want to use for trigger condition to the *Variable Mappings* list. You can take any signal from the Variable browser or a signal directly from a plotter.
   TRY:
   Drag the signal generator output to the plotter, take the output of the block with the name u1. If you drop it at the y-axis of an existing signal in the plotter they will share y-axis, if you drop it anywhere else it will get a new y-axis. Use the signalgenerator output as trigger signal for the plotter as described above.
4. The easiest way to zoom in to a plot is to stop the measuring, (blue icon), grab the scrollbar under the plot and resize it by grabbing it at one of its ends. Then you can then scroll through your plot. If you zoom manually with the mouse in the plot and want to zoom out again you have to right click the plotter and deselect zoomed view. It is also possible to hit the U character on tkeyboard.
5. Check if the motors rotates in correct direction, i.e., positive voltage should give positive velocity and vice versa. Some motors turns in the wrong direction, this can be corrected in the model, by multiplying the input to the PWM block with a gain block and enter minus one (-1.0).

Saving signals from ControlDesk to Matlab

The ControlDesk plots can be exported to Matlab where they can be manipulated and printed and/or exported to a word processing program etc.
Stop the measuring and right click in the plotter, select *Save Displayed Data as New Measurement.* Open the *Project* window, in the *Measurement data directory* you have a *rec001.idf* file, double click it and then right click it, select export and file format .mat. In

Matlab execute, >> load rec001. Since the variable rec001 has a structure you have to find the x and y data in the structure. Plot the signal to see that it is the same as in ControlDesk.

Saving and backing up the ControlDesk project

There are dependencies between files in a project which means that you can not just copy a project from one directory or computer to another.

First save the project + Experiment then take a Backup of the Project + Experiment. Now you have a zip file with the ControlDesk files that you can open on any one of the computers from Open Project + Experiment from Backup.

--*End of the tutorial, return to the workshop A document*

Connecting more than one block parameter to one instrument

This may have to be done later depending on how you have designed your Simulink file.

1. Create an Instrument and connect the parameter of the first Simulink block that you would like to change to the instrument.
2. Connect the second parameter from the second Simulink block by dragging it with the right mouse button instead of the left. You should then get a question of how you would like the second connection to be configured. Choose *Connect as additional Write Variable*.

Online help

The online help in ControlDesk is called Help Desk (Help -> Contents) and is an Explorer based help browser, it is simple to use. There is much functionality in ControlDesk which is not described in this document, so please use the help if you need some special functionality of ControlDesk.