Aero 2 Lab Procedure
Rotor Step Response Modelling

Setup

1. Make sure the Aero 2 has been tested as instructed in the Quick Start Guide.
2. Launch MATLAB and browse to the working directory that includes the Simulink models for this lab.
3. Make sure the Quanser Aero is configured as the 1 DOF pitch-only system:
    a. Unlock the pitch axis and lock the yaw axis.
    b. Both rotors are horizontal.
    c. Mount weight on each rotor.
4. Connect the USB cable to your PC/laptop.
5. Connect the power and turn the power switch ON. The Aero base LED should be red.

Capturing Rotor Speed Response

The *q_aero2_rotor_step* Simulink model shown in Figure 1 applies a 10V step to the front rotor, i.e. motor 0, for 5 sec and outputs the measured rotor speed using the digital tachometer.

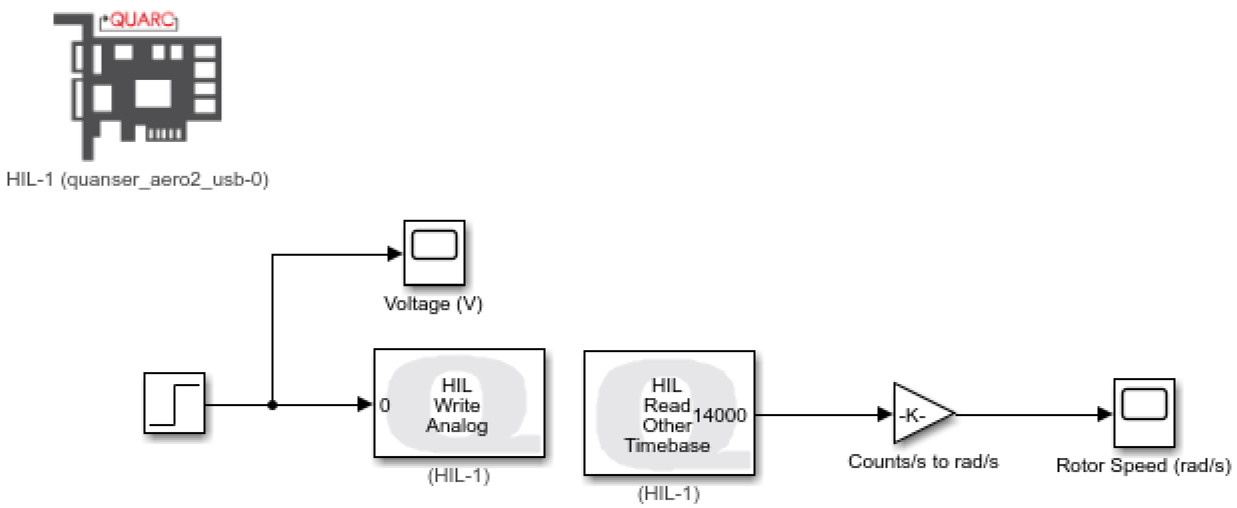

Figure 1 - q_aero2_rotor_step SImulink model.

Follow the instructions below to apply a step and measure the rotor response:

1. Open the *aero2_rotor_step_response_modelling_student.mlx* MATLAB Live Script.
2. Open the the *q_aero2_rotor_step* Simulink model.

3. Build and run the *q_aero2_rotor_step* Simulink model in QUARC to apply the step to the front rotor, i.e., motor 0. To do this, click on the *Monitor & Tune* button in the Simulink menu under the *Hardware* tab. The model will stop automatically after 5 sec. The response should look similar as shown in Figure 2.

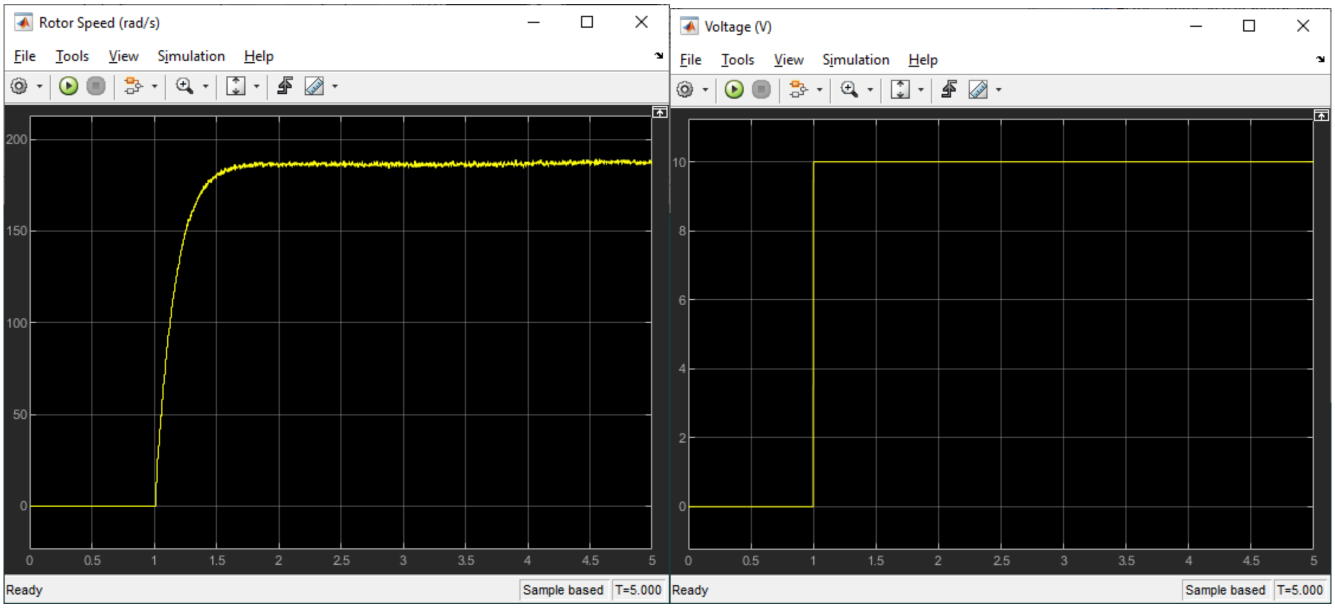

Figure 2 - Rotor speed response

4. Plot the rotor speed response and motor voltage in a MATLAB figure. The motor voltage and rotor angular speed are saved in the MATLAB variables *DataRotorVm* and *DataRotorSpeed*. See the example code in the Live Script. Attach the response.

5. Close the Simulink model

Find the Transfer Function

1. Find the steady-state gain, K, of the system from the measured response.

   Hint: You can use the *Cursor Measurements* tool in the Simulink Scopes or the take data points directly from the MATLAB figure to take your measurements.

2. Find the time constant, τ, of the system using the measured response.

3. What is the resulting transfer function model of the rotor?

Model Validation

In this section, the rotor model you found is ran in parallel with the hardware to see how well the model represents the actual system. The *q_aero2_rotor_model_val* Simulink model shown in Figure 3 applies a 10V step to the Aero 2 front rotor, i.e., motor 0, and the transfer function model. The rotor speed measured from the digital tachometer and the simulated rotor speed from the model are displayed in the *Rotor Speed* scope.

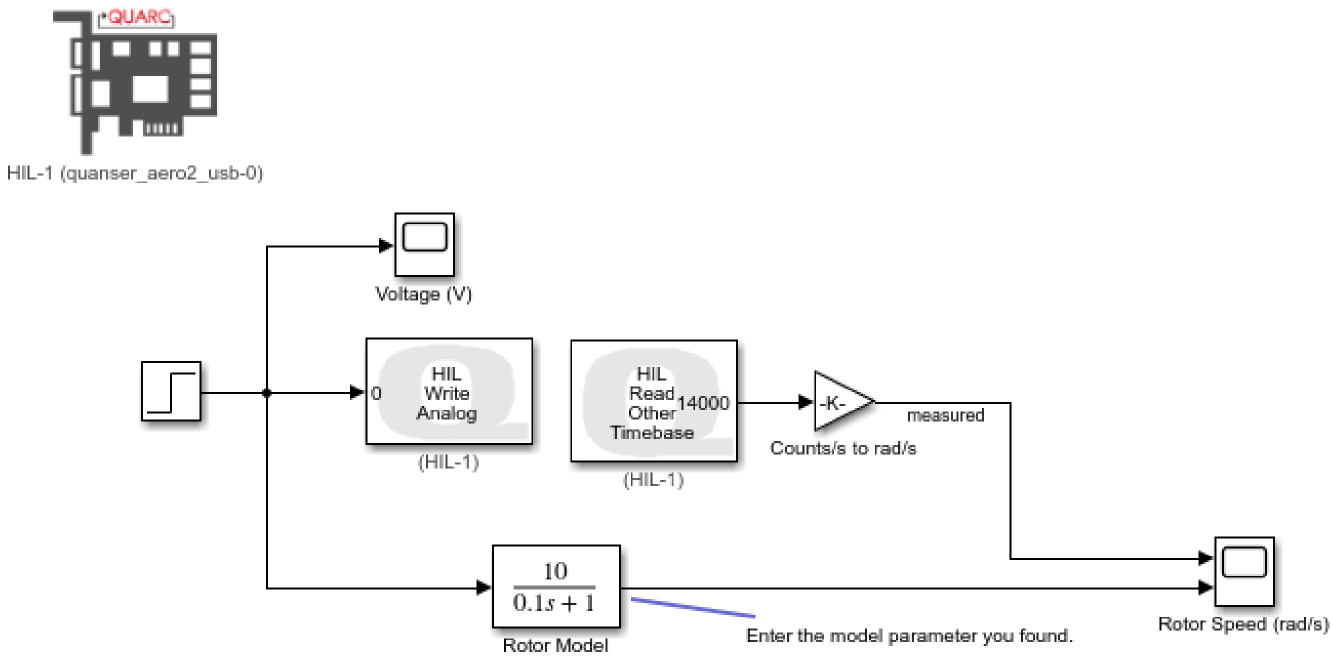

Figure 3 - q_aero2_rotor_model_val Simulink model is used to validate the rotor model.

Follow these instructions to compare the model with the hardware response:

1. See the *Model Validation* section in the Live Script.
2. Open the *q_aero2_rotor_model_val* Simulink model.
3. Enter the transfer function parameters, K and τ, that you found in the *Rotor Model* Transfer Fcn block in the Simulink model, as illustrated in Figure 3.
4. Build and run the *q_aero2_rotor_model_val* Simulink model in QUARC by clicking on the *Monitor & Tune* button in the Simulink *Hardware* tab. See the sample response shown in Figure 4. Note that this is using the default transfer function $10/(0.1s + 1)$ shown in Figure 3 and is not the correct model of the rotor.

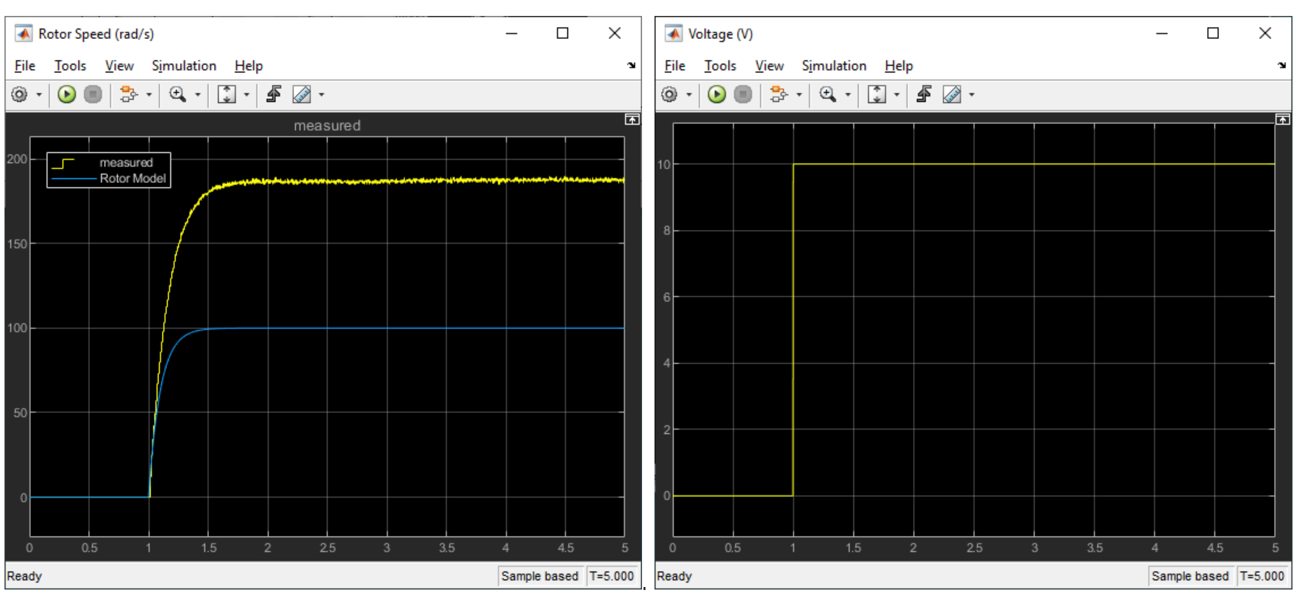

Figure 4 - Sample response from q_aero2_rotor_model_val Simulink model.

5. Attach a MATLAB figure showing the response of the rotor speed of both the hardware and the model and the motor rotor voltage. See the example code in the Live Script to create this plot.
6. Does your model represent the Aero 2 well? Explain.
7. **[Optional]** Is the rotor system stable? Explain using both the model and from your observation running the step response lab.
8. Close the Simulink model
9. Turn off the power on the Aero 2.