

# ECE-320 Lab 7: Discrete-Time PID and PI Controllers and sisotool

## Overview

In this lab you will be controlling both of the one degree of freedom systems you previously modeled using discrete-time PID and PI controllers. Both one degree of freedom systems must be controlled, and if there are two people in your lab group each lab partner should do a different system.

You will need to download the files for Lab 7 from the class website.

**Design Specifications:** For each of your systems, you should try and adjust your parameters until you have achieved the following:

### Rectilinear Systems (Model 210)

- Settling time less than 1.5 seconds.
- Steady state error less than 0.1 cm for a 1 cm step, and less than 0.05 cm for a 0.5 cm step
- Percent Overshoot less than 25%

Your memo should include **four** graphs **for each** of the 1 dof systems you used (a PID controller and a PI controller for each system, two different sampling intervals). Your memo should compare the difference between the predicted response (from the model) and the real response (from the real system) for each of the systems.

### Background/Review

The file **DT\_PID.mdl** is a Simulink model that implements a discrete-time PID controller. It is somewhat unusual in that the plant is represented in state-variable form, but this is the usual form we will use in this class. The Simulink model looks like the model shown in Figure 1:

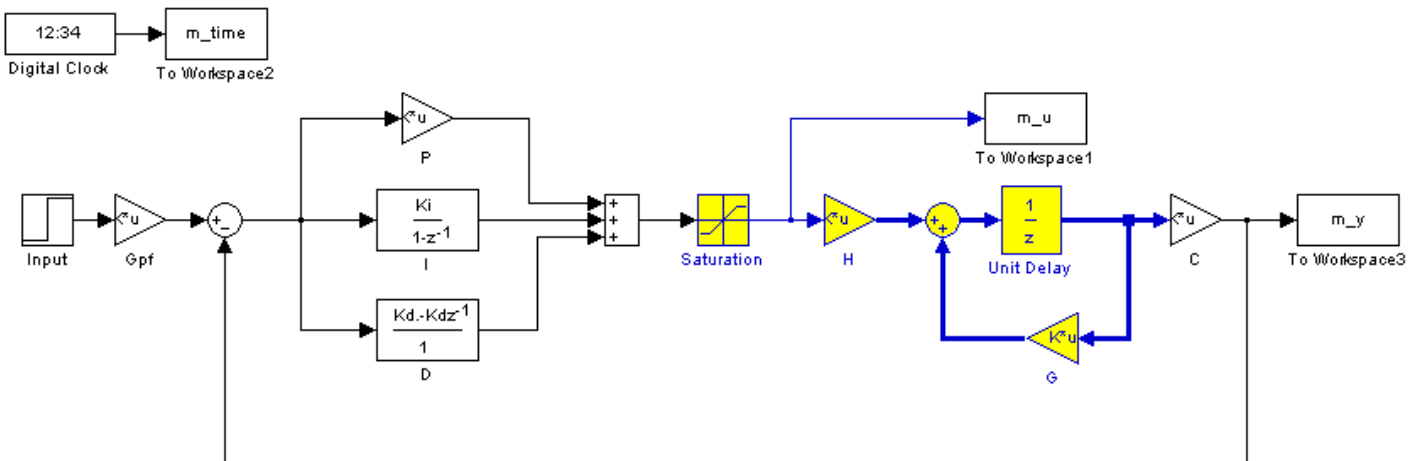


Figure 1. Simple feedback loop with a discrete-time PID controller.

The file **DT\_PID\_driver.m** is the Matlab file that runs this code. We will be utilizing Matlab's *sisotool* for determining the pole placement and the values of the gains.

Before we go on, we need to remember the following two things about discrete-time systems:

- For stability, **all poles of the system must be within the unit circle**. However, zeros can be outside of the unit circle.
- The closer to the origin your dominant poles are, the faster your system will respond. However, the control effort will generally be larger.

The basic transfer function form of the components of a discrete-time PID controller are as follows:

Proportional (P) term :  $C(z) = K_p E(z)$

Integral (I) term:  $C(z) = \frac{K_i}{1-z^{-1}} = \frac{K_i z}{z-1}$

Derivative (D) term :  $C(z) = K_d(1-z^{-1}) = \frac{K_d(z-1)}{z}$

**PI Controller:** To construct a PI controller, we add the P and I controllers together to get the overall transfer function:

$$C(z) = K_p + \frac{K_i z}{z-1} = \frac{(K_p + K_i)z - K_p}{z-1}$$

In *sisotool* this will be represented as

$$C(z) = \frac{K(z^2 + az)}{z(z-1)} = \frac{K(z+a)}{(z-1)}$$

In order to get the coefficients we need out of the *sisotool* format we equate coefficients to get:

$$K_p = -Ka, \quad K_i = K - K_p$$

**PID Controller:** To construct a PID controller, we add the P, I, and D controllers together to get the overall transfer function:

$$C(z) = K_p + \frac{K_i z}{z-1} + \frac{K_d(z-1)}{z} = \frac{K_p z(z-1) + K_i z^2 - K_d(z-1)^2}{z(z-1)} = \frac{(K_p + K_i + K_d)z^2 + (-K_p - 2K_d)z + K_d}{z(z-1)}$$

In *sisotool* this will be represented as

$$C(z) = \frac{K(z^2 + az + b)}{z(z-1)}$$

In order to get the coefficients we need out of the *sisotool* format we equate coefficients to get:

$$K_d = Kb, \quad K_p = -Ka - 2K_d, \quad K_i = K - K_p - K_d$$

For the PID controller, we can have either two complex conjugate zeros or two real zeros.

## *Sisotool (Brief) Example*

Run the Matlab program *DT\_PID\_driver.m*. This program is set up to read the data file *bobs\_210\_model.mat*, which is a continuous time state variable model for a one degree of freedom rectilinear system, construct an equivalent discrete-time system using a sample and hold with a delay (the sample time is given by  $T_s$ ), and implement a  $P$  controller with gain 0.0116. It will put the value of the transfer function for your system,  $G_p(z)$ , in your workspace. Now we are ready for *sisotool*.

### *Getting Started*

- Type **sisotool** in the command window
- Click **close** when the help window comes up
- Click on **View**, then **Design Plots Configuration**, and turn off all plots except the **Root Locus** plot (set the **Plot Type** to **Root Locus** for **Plot 1**, and set the **Plot Type** to **None** for all other Plots)

### *Loading the Transfer Function*

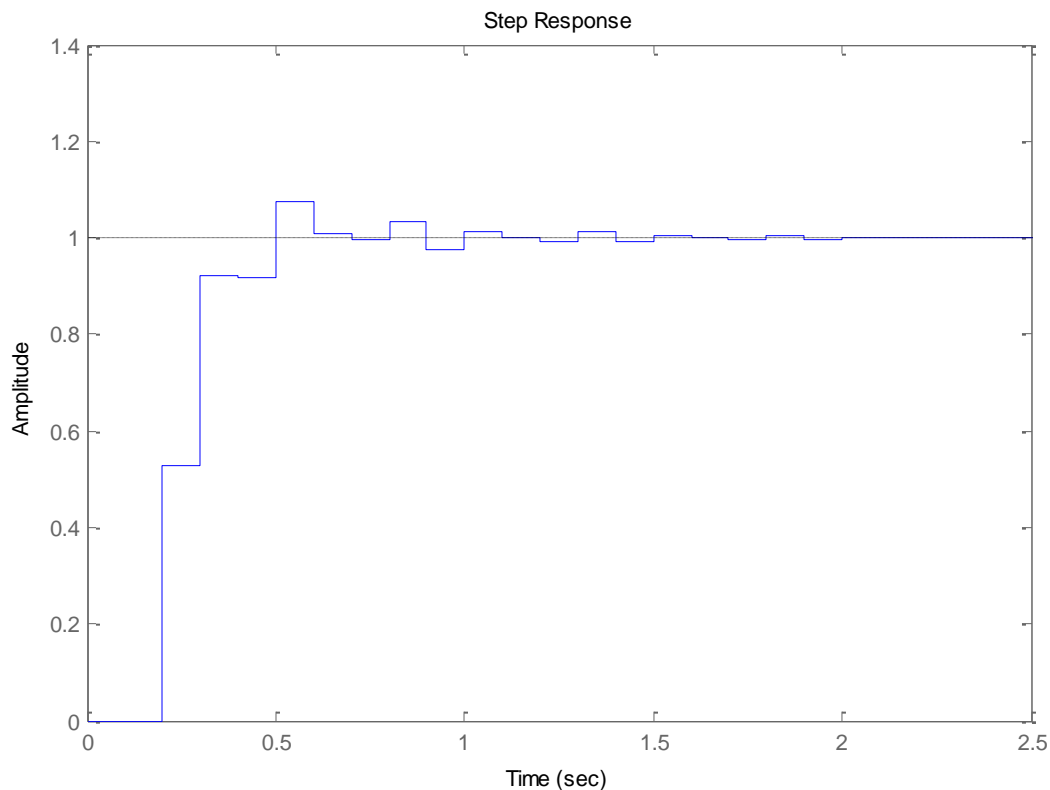
- In the **SISO Design** window, Click on **file** → **import**.
- We will usually be assigning  $G_p(z)$  to block  $G$  (the plant). Under the **System** heading, click on the line that indicates **G**, then click on **Browse**.
- Choose the available Model that you want assigned to  $G$  (Click on the appropriate line) and then click on **Import**, and then on **Close**.
- Click **OK** on the System Data (Import Model) window
- Once the transfer function has been entered, the root locus is displayed. Make sure the poles and zeros of your plant are where you think they should be. Note that there will be three poles (one at zero) for our second order system since for this system there is a delay inserted by the Simulink model.

### *Generating the Step Response*

- Click on **Analysis** → **Response to Step Command** (the system is unstable at this point)
- You will probably have two curves on your step response plot. To just get the output, type **Analysis** → **Other Loop Responses**. If you only want the output, then only  $r$  to  $y$  is checked, and then click **OK**. However, sometimes you will also want the  $r$  to  $u$  output, since it shows the control effort for  $P$ ,  $I$ , and  $PI$  controllers.
- You can move the location of the pole in the root locus plot by putting the cursor over the pink button and holding the left mouse button down as you move the pole locations. You should note that the step response changes as the pole locations change.
- The bottom of the root locus window will show you the closed loop poles corresponding to the cursor location if you hold down the left mouse button. However, if you need all of the closed loop poles you have to look at all of the branches.

Entering a Compensator (controller): We will implement a PI controller here

- Click on **Designs**, then **Edit Compensators**.
- Right click in the **Dynamics** window to enter real poles and zeros. You will be able to change these values very easily later. Since we want a PI controller, we need a pole to be a 1 and we need to be able to change the value of the zero. For now assume the zero is at -1.
- Look at the form of  $C$  to be sure it's what you intended, and then look at the root locus with the compensator.
- You can again see how the step response changes with the compensator by moving the locations of the zero (grab the pink dot and slide it) and moving the gain of the system (grab the squares and drag them). Remember we need all poles and zeros to be inside the unit circle for stability!
- Move the pole and zero around until the zero is approximately -0.295 and the gain is approximately 0.0563. You should get a figure like that shown in Figure 2.



**Figure 2.** Discrete-time example with w PI controller.

*Adding Constraints*

- Right Click on the Root Locus plot, and choose **Design Requirements** then either **New** to add new constraints, or **Edit** to edit existing constraints.
- At this point you have a choice of various types of constraints.
- *Remember these constraints are only exact for ideal second order systems!!!!*

*Printing/Saving the Figures:*

To save a figure **sisotool** has created, click **File** → **Print to Figure**

*Odds and Ends :*

You may want to fix the axes. To do this,

- Right click on the Root Locus Plot
- Choose **Properties**
- Choose **Limits**
- Set the limits and turn the **Auto Scale** off

You may also want to put on a grid, as another method of checking your answers. To do this, right click on the Root Locus plot, then choose **Grid**

It is easiest if you use the zero/pole/gain format for the compensators. To do this click on **Edit** → **SISO Tool Preferences** → **Options** and click on **zero/pole/gain**.

*Back to Matlab.*

- Determine the correct values of  $a$  and  $K$
- Enter these in the Matlab code **DT\_PID\_driver.m**
- Modify **DT\_PID\_driver.m** to compute the proportional and integral gains
- Run **DT\_PID\_driver.m** and print out the picture and attach it to this homework. It should look like Figure 1.

For each of your two 1 dof systems, you will need to go through the following steps:

**Step 1:** Set up the 1 dof system exactly the way it was when you determined its model parameters.

**Step 2:** Modify **DT\_PID\_driver.m** to read in the correct model file. You may have to copy this model file to the current folder.

**Step 3:** Modify **DT\_PID\_driver.m** to use the correct *saturation\_level* for the system you are using.

**Step 4:** Set the sampling interval to 0.1 seconds (the first time) and then 0.05 seconds (the second time).

**Step 5:** PID and PI Control

- Design a PID and then a PI controller using *sisotool* to meet the design specs. Use a **constant prefilter** (i.e., a number, most likely the number 1)
- Implement the correct gains into **DT\_PID\_driver.m**
- Simulate the system for 2.0 seconds. If the design constraints are not met, or the control effort hits a limit, redesign your controller (you might also try a lower input signal). Try and stay away from the maximum allowed control values as much as possible, they are not as good a predictor with discrete-time systems as with continuous time systems.
- Reset the system using **ECPDSPReset.mdl**
- Compile the correct closed loop ECP Simulink driver (**Model210\_DT\_PID.mdl**), connect to the system, and run the simulation.
- Use the **Compare\_DT1.m** file (or a modification of it) to plot the results of both the simulation and the real system on one nice, neatly labeled graph. You need to include this graph in your memo.