Lab Session 6
Lead Controller Design

By: Professor Dan Block
Control Systems Lab Manager
University of Illinois

Equipment

Agilent 54600B 100 MHz Ditizing Oscilloscope
Agilent 33120A Function/Arbitrary Waveform Generator
Agilent 6632A Power Supply
Agilent 34401A 6 1/2 Digit DMM
Agilent 35670A Dynamic Signal Analyzer
Agilent VEE 5.0 software
Matlab software (www.mathworks.com)
Amplifier/Motor/Potentiometer test set (hybrid, built in-house)
Analog computer: Comdyna GP6

Purpose

In Lab Session 5, motor position and angular velocity measurements were used to construct a compensator. Frequently in practice, derivative measurements are not directly available, and differentiation of position measurements may not be advisable because of measurement noise. Lead compensators are a class of controllers that can be successfully used even when derivative information is not directly available. Lead compensators have the general form

$$G_c(s) = K_c \frac{1+s\tau_z}{1+s\tau_p},$$

with $\tau_p < \tau_z$. The pole provides high frequency gain roll-off that results in less amplification of high frequency measurement noise than is observed with a PD compensator. This is one significant advantage of the lead compensator. In this set of experiments, you will build upon the frequency domain viewpoint which was introduced in Lab Session 4. In this lab you will directly use a Bode plot rather than a root locus to design the lead compensator G_c to meet the following objectives:

- Reasonable bandwidth;
- Good transient response;
- Good disturbance rejection to overcome motor friction.

Preparation

Readings:

From Kuo,

- Frequency Response and Bode plots, Secs. A-2, 9-1, 9-3.
- Stability Margins, Secs. 9-14, 9-15.
- Lead Compensation, Sec. 10-5.

Prelab:

(a) First we consider proportional control, where $G_c(s) = K_c$. Develop a Bode plot of the resulting loop transfer function

$$G(s) = V_\theta(s) / V_i(s) = K_c \frac{1}{s} \frac{K_{pot} K_{amp} K}{1 + s\tau_m}$$

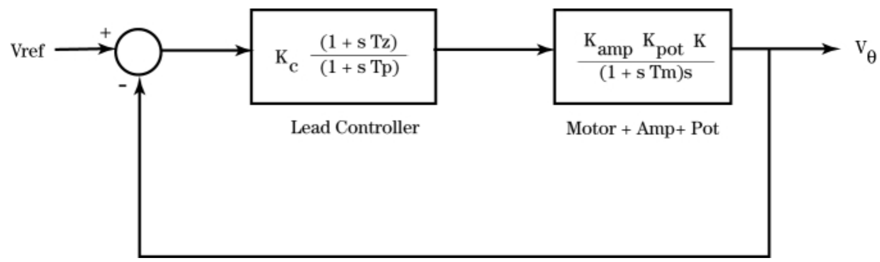

Figure 6. 1. DC Motor Lead Controller System.

Use K_{pot}= 10/(2π), K = 18, τ_m = 1/10, K_{amp} **=** 2.4 and K_c = 1.

To obtain a short rise time, the closed loop bandwidth must be correspondingly large. A good rule of thumb for design is to approximate the closed-loop bandwidth by the open-loop crossover frequency since we typically have $\omega_n \leq \omega_{bw} \leq 2\omega_n$. We will use this heuristic to approximate the closed-loop bandwidth in our control designs. Using your Bode plot obtained above, determine the crossover frequency ω_c of the system when no compensation is used (i.e. K_c = 1), and find the phase margin.

Now determine the necessary value of K_c to achieve ω_c = 55 rad/s. Plot the magnitude and phase response of this compensated open loop transfer function, and determine the phase margin (PM) of your design. For a second order system, as the PM approaches O°, the damping ratio ζ is reduced and the system response becomes more oscillatory. Do you notice a decrease in the phase margin as you increase the gain?

To improve the PM you will now design a lead compensator.

(b) Obtain a Bode plot for the lead compensator with transfer function

$$G_c\,(s) = K_c \left(\frac{1+s0.1}{1+s0.01}\right).$$

You may take K_c = 1. Note that the phase plot has a positive bulge in the frequency range 3 rad/s $\leq \omega \leq$ 400 rad/s. When a lead compensator such as this is cascaded with the motor, their phase responses add to increase the overall phase in this frequency band. In your design, you will need to choose the size and frequency of the center of this phase bulge.

(c) A lead compensator plus motor gives the loop transfer function:

$$G_c(s)\,G(s) = K_c \left(\frac{1+s\tau_z}{1+s\tau_p}\right)\left(\frac{K_{pot}\,K_{amp}\,K}{s\,(1+s\tau_m)}\right)$$

The design in Lab 5 had approximately 15% overshoot ($M_p \approx$ 0.15) and approximately 20 ms rise time ($t_r \approx$ 20ms). Use the formulas from Kuo, as referred to in Session 1, to get the necessary ζ and ω_n. These pole location specifications for a second-order system can be translated to frequency specifications on an open-loop system—see Kuo Sec. 7-5.

Find the necessary PM and ω_c as though we had a second order system. This will serve as a starting point for your design. Using frequency domain techniques, design G_c to obtain this cross over frequency and phase margin, and a dc gain $2 \le K_c \le 4$. As before, we are interested in the magnitude and phase characteristics of the open-loop response to determine the closed-loop response, using the feedback configuration illustrated in Fig. 6. 1.

Give the resulting closed loop system, and plot its step response. Check that you met the time-domain specifications and adjust your design if necessary. Document the choices you made for each iteration. Obtain a root locus plot of your final design with K_c varied from 0 to ∞, and on the root locus mark the closed loop pole locations for your design.

(d) Consider a lead controller with transfer function

$$G_c(s) = 0.30\frac{s/2.3+1}{s/92+1} \tag{6.1}$$

Verify that this controller meets the PM and cross over frequency requirements. Plot the step response of the closed loop system, and verify the time domain specifications also. Note however, that the dc gain of this controller is only 0.14, which should be much smaller than the compensator that you designed. Theoretically both designs should produce a good step response, and since we have a type 1 system, we should obtain zero steady state error. However, this argument ignores the affects of disturbances: In practice we will find that a larger dc gain is required to overcome the effect of Coulomb friction.

(e) The lead controllers will be implemented using the wiring diagram shown in Figure 6.2. Analyze this diagram and show that the potentiometer settings P_1, P_2 and P_3 are related to the controller parameters K_c, τ_z, and τ_p by the set of Equations (6.2) below.

Laboratory Excercise

Step Response with Low DC Gain Lead Controller:

The object of this exercise is to study the step response of the motor with the low dc gain controller (6.1) that you have studied in the prelab. Since this is a type 1 system, in the absence of disturbances the steady state error will be zero. In this experiment you will see that this compensator is not able to overcome the Coulomb friction of the motor. This will result in a large steady state error in the step response. The compensator will be implemented using the analog computer.

- The waveform generator should have the following settings.

  1. Output: Hi Z. To set the expected load impedance, follow these steps: Press the **Shift** key and the **Enter** key (the one with **Menu** written in black above it). Press the > key 3 times until the menu item displayed on the screen is *Sys Menu.*

     Press the V key twice until the display reads *Parameter/ 50 ohms*. Now press the > key until you see *High Z* displayed on the screen. At this point press the **Enter** key.

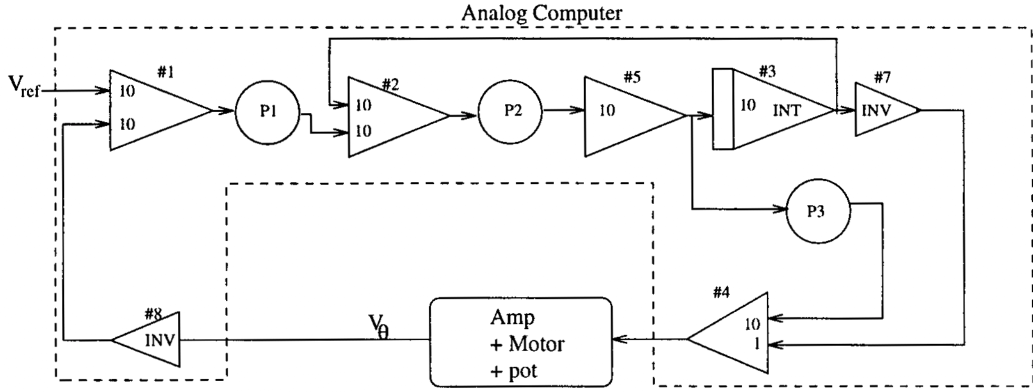

Figure 6.2. Wiring diagram for motor with lead controller

2. Frequency = 200 mHz

3. Amplitude = 1 V p-p

4. Waveform = square

5. DC offset = 2 V

Connect the waveform generator to amplifier #1 on the analog computer as shown in Figure 6.2.

- Connect the +10 V on the patch panel to the bottom-most potentiometer jack—This is the potentiometer that is used to measure motor output. Connect the black jack corresponding to +10V to the top-most potentiometer jack. Connect the top-most potentiometer lead to the analog computer ground and the middle lead to inverter 8's input.

- Wire the analog computer as shown in Figure 6.2. The output of the amplifier #4 on the analog computer should be connected to the **Input Amplifier** on the patch panel. To do this, take one end of the grey cable (the one with three leads on each end) and connect the red lead to the analog computer output. Connect the black and white leads to the analog computer ground. Take the other end of the grey cable and connect the red lead to Ref+ of the **Input Amplifier.** Connect the black and white leads to the black and white jacks of the Input Amplifier. Connect the motor to the **Output Amplifier** by using the other grey cable. Short out the unused amplifier #6.

- The relation between the three potentiometer settings and the controller parameters is as follows.

$$P_1 = Kc/10$$
$$P_2 = 1/(1000 * \tau_p) \qquad (6.2)$$
$$P_3 = \tau_z$$

  Turn on the analog computer, and set the potentiometers to the parameter values corresponding to the lead controller (6.1).

- Connect the cable attached to the push button to **Amp Inhibit** on the patch panel.

- The scope settings are:

  1. Channel 1 and Channel 2: Both should be On, and should be set on D.C. Coupling, 500mV/div. Ground line one division up from the bottom of the screen.
  2. **Main mode**
  3. **Timebase =** 50 ms
  4. **Trig. Mode =** Normal
  5. **Trig. Level =** 2 V
  6. **Trig. Source =** Ch 2

- Connect Channel 1 of the scope to the middle lead of the potentiometer and its ground to the topmost potentiometer lead.
- Connect Channel 2 of the scope to the waveform generator. This connection can be made at the input to Amplifier #1 on the analog computer. Connect the ground to the analog computer ground. You should be able to observe the input square wave on the scope.

Before you proceed, have a TA check your wiring.

- Turn on the amplifier. Put the analog computer in OPR mode. Depress the push button and keep it depressed. The motor should respond to the step input, and you should be able to see the potentiometer output on the scope. If the motor starts spinning wildly, release the push button and check your connections, particularly those on the analog computer. If the motor does not respond, try moving the flywheel closer to the reference signal on the scope.
- Using the scope measurement functions and the cursors, measure the output amplitude, overshoot, rise time, settling time, and the steady state error. Does the position of the flywheel before the step make a difference?
- Note that the step response has a steady state error. In between transitions of the square wave, physically rotate the flywheel with your fingers and observe if the controller is able to bring the motor back to its previous position. The control should feel *loose,* indicating a low dc gain. Measure the steady state error.

Step Response with Greater DC Gain Compensation:

In the previous exercise, you should have seen that the controller was weak and not able to overcome disturbances. We shall see in this exercise that a compensator with greater dc gain is useful in overcoming the affect of disturbances.

- Retain all the connections that you had in the previous lab exercise.
- Change the potentiometer settings P_1, P_2 and P_3 to correspond to the controller parameter values that you designed in the prelab, in the same way that you did in the previous exercise.
- Put the analog computer in OPR mode. Depress the push button and keep it depressed.
- Note that the step response no longer has a steady state error. In between transitions of the square wave, physically rotate the flywheel with your fingers and observe if the controller is able to bring the motor back to its previous position. The control should feel much *tighter* than before, indicating a higher dc gain.
- Observe the step response and measure the output amplitude, overshoot, settling time and rise time.

Phase Margin Measurement for the High DC Gain Design:

The object of this exercise is to measure the phase margin obtained by the use of the lead controller. Phase margin is a measure of stability of a system. For a second order system the damping ratio $\zeta \approx$ *PM in degrees*/100. In this exercise, we will use the high dc gain lead controller. The phase margin is obtained by studying the open loop frequency response. In this exercise, we will study the frequency response of

$$\frac{V_{tach}}{V_i}(s) = \frac{(K_c/10)(1+\tau_z s)}{1+\tau_p s}\frac{KK_{tach}K_{amp}}{1+s\tau_m} \quad (6.3)$$

(A controller gain of K_c/10 instead of K_c will be used to avoid saturating Amplifier#l in the analog computer.) We however, are interested in the frequency response of

$$\frac{V_\theta}{V_i}(s) = \frac{K_c(1+\tau_z s)}{1+\tau_p s}\frac{K}{1+s\tau_m}\frac{K_{pot}K_{amp}}{s} \quad (6.4)$$

You will make the necessary transformations to the samples of the frequency response as part of your report.

- Turn off the analog computer, the motor amplifier and the waveform generator. Turn on the DSA.
- Change the **Timebase** on the scope to be 500 ms. Switch to **ROLL** mode. Use the Main/Delayed button for this. The other scope settings are the same as before.
- Be sure the potentiometer settings are correct for the high dc gain controller.
- The wiring diagram is shown in figure 6.3.

It is identical to the wiring diagram in the previous exercise except for the following:

1. Remove all electrical connections to and from the motor potentiometer. Disconnect it from the motor assembly.
2. Remove the connection between the inverter #8 and amplifier #1 on the analog computer.
3. The input is now provided by a source on the DSA.
4. The input voltage now gets multiplied by 1 and not by 10, by amplifier #1.

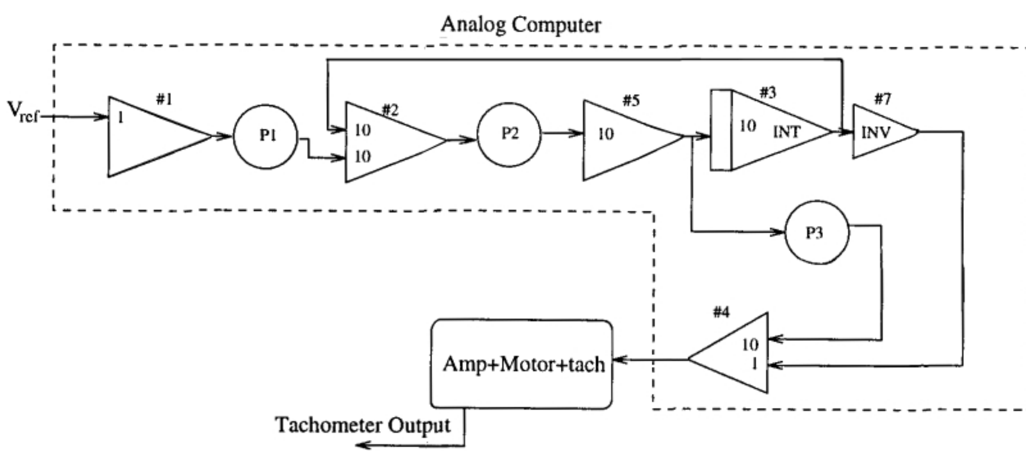

Figure 6.3. Wiring diagram for frequency response.

Make the following connections.

- Connect Channel 1 of the DSA (the input) to the input of Amp#l on the analog computer.
- Connect Channel 1 of the scope to the input from the DSA.
- Connect Channel 2 of the scope to the tachometer's orange jack and its GND to the tachometer's grey jack.
- Connect Channel 2 of the DSA to the tachometer output.

Have a TA check your wiring

We will use AgilentVee to control the DSA. Open the file: `f:\labs\Ee386\dsacontrol.vee`. This will use the DSA to get samples of the frequency response from Channel 1 to Channel 2.

- Press the **Initialize** button in AgilentVee.
- Turn on the analog computer and the amplifier.
- Put the analog computer in OPR mode and insert the $\frac{1}{4}$ inch plug into the pushbutton jack.
- Press **Start Measurement** in AgilentVee.
- Enter 0.5 for the start frequency, 20 for the stop frequency, 15 for the number of points, 1.5 for the magnitude, and 8 for the offset voltage.

Watch the waveforms on the DSA and the scope. When the sine sweep is completed (takes a few minutes), you must save your data to disk.

- Pull out the $\frac{1}{4}$ inch plug from the pushbutton jack.
- Press **Collect Data.** The frequency response samples will be saved into file `a:\fresp.m`.
- Press **Curve Fit.** Enter the frequency range for the fit—usually the same as the sweep, unless the high frequency data looks corrupted. Enter 2 for the number of poles and 1 for the number of zeros. The transfer function will be saved to disk in another file, `a:\frespf.m`.

You will run these script files in MATLAB to find the PM of the system specified in Equation 6.4.

Report:

1. Calculate the motor dc gain from the DSA transfer function fit's dc gain (remember to remove K_{tach}, $K_C/10$, and K_{amp}), and compare it with the values obtained in Experiments 4 and 5.
2. For the two controllers used, compare the overshoot, rise time, settling time and steady state error with the values that you would expect based on your model (you must simulate the system). Look at the error in the model of the motor parameters, the dc gain and the slow pole in the DSA fit. Redo the simulation with with these values for the motor model. Discuss.
3. Comment on the two controllers, and compare them with the PD controller used in Experiment 5. Could you in each case meet the specs on the real motor?

4. Use MATLAB to make a Bode plot of the transfer function fit by the DSA and overlay (magnitude and phase) the frequency samples of $\frac{V_{tach}}{V_i}(s)$ obtained from the data in fresp.m (use a different line-type). Note the quality of the fit.

5. Obtain the frequency response of $\frac{V_\theta}{V_i}(s)$ from the poles and zeros fit by the DSA, using the relations between Equations 6.4 and 6.3, and make a Bode plot. Overlay as above after computing samples of $\frac{V_\theta}{V_i}(s)$ from the data in fresp.m. Find the phase margin and cross-over frequency ω_C.

6. Use this open-loop transfer-function-fit to compute the closed-loop transfer function and make a Bode plot with MATLAB. Compute samples of the closed loop response from the samples of $\frac{V_\theta}{V_i}(s)$ and overlay. What is the closed-loop bandwidth?

Step Response Data Table:

	Low DC Gain	High DC Gain
V_{amp}		
M_p		
t_r		
t_s		
e_{ss}		
“Stiffness”		