

1. At the Matlab prompt, type `simulink` to start Simulink.
2. Click on the white “new document” tab in the upper lefthand corner of Simulink Library Browser to open a Simulink workspace.
3. In the Simulink Library Browser click on the plus sign to look under the *Simulink* directory. Click on the word *Continuous*. Click and drag a *Transfer Fcn* block to the Simulink workspace (the window labeled *Untitled*). Then click and drag a second transfer function to the workspace.
4. If you select a block (left click on it) and enter *CTRL R* that will rotate the block. This is useful for putting blocks in the feedback path.
5. In the Simulink Library Browser, click on the word *Math*. Click and drag an *Addition* or *Sum* block to the workspace. You can also find a *Gain* block here.
6. In the Simulink Library Browser, click on the word *Sources*. Click and drag two *Step* blocks to the workspace; Click on the word *Sinks* and drag a *Scope* block to the Simulink workspace.
7. Connect the blocks together by clicking on an output tab of one block and dragging a line to the input tab of another block.
8. In your Simulink workspace, double click on the step block; change the *Step Time* (starting time) to be zero for a step input starting at time zero. The default start time for a step block is 1 second. Double click on the addition or sum block; change the second sign to be negative.  
  
Double click on a transfer function block; change the numerator and denominator entries to be the desired polynomials, where the polynomial coefficients are written from the highest power of  $s$  down to the constant term. For example, the polynomial  $s + 2$  would be entered as `[1 2]`; the polynomial  $s^3$  would be entered as `[1 0 0]`. Also, change the label on this block to be *Compensator*.  
  
Connect the *Scope* block to the plant output.
9. Click on the play button (black triangle) to start the simulation. The number to the right of the play button determines how long (in seconds) the simulation will go.
10. Click the binoculars on the scope blocks to autoscale the plots.
11. If you want to export data from a Simulink scope to Matlab, do the following. Click on the Parameters tab of the scope (this is the second tab, to the right of the printer icon). Click on the Data History tab. Uncheck the box that says Limit data points to last 5000. Check the box that says Save data to workspace. Type in a variable name. Change the format to Array. This will save the variable as an array (matrix) with two columns. The first column contains time values and the second column contains the corresponding variable values at those times. For example, if the variable is called  $y$ , you can plot it in Matlab as follows: `>>plot(y(:,1),y(:,2));grid.`