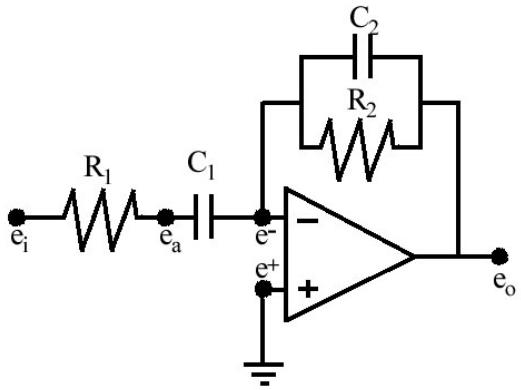
1 Objective

The objective of this tutorial is to model a band-pass filter consisting of an op-amp with a series combination of a resistor and capacitor in the feed-forward path and a parallel combination of a resistor and a capacitor in the feed-back path.

2 Quick Analysis



From the State Space Model (a subject in Contril system), the governing **differential equations** for e_{C1}' and e_{C2}' were found to be:

$$e_{C1}' = -\frac{1}{C_1 R_1} e_{C1} + \frac{1}{C_1 R_1} e_i$$

$$e_{C2}' = -\frac{1}{C_2 R_1} e_{C1} - \frac{1}{C_2 R_2} e_{C2} + \frac{1}{C_2 R_1} e_i$$

Using this model we will find the step response of this system and perform a frequency sweep.

2 Creating the Model

To begin modeling,

- start Simulink
- open a new model
- save the model as ***opamp.mdl.***

This op-amp system is governed by two differential equations. Each differential equation expresses the first derivative of one of the state variables in terms of the states of the system and the system input.

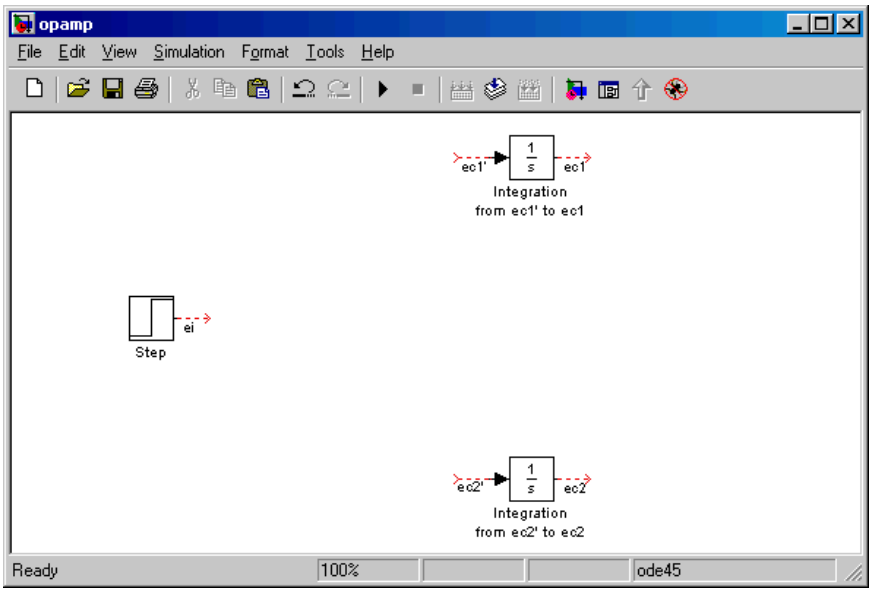
The first step is to create relations between e_{C1} and e_{C2} and their derivatives.

In the Simulink Library Browser

- expand Simulink.
- single-click on Continuous.
- drag two Integrator blocks into the Simulink model window.

In the Simulink Library Browser window,

- expand Simulink,
- click on Sources,
- drag the Step source  from the right column into the model



- Simulink Library Browser → Math Operations category window → Gain block
- right-click on the Gain block
- select Format → click on Flip block
- double-clicking on the Gain block → change the gain from 1 to 1/(C1*R1)
- Enlarge the gain block

Similarly add

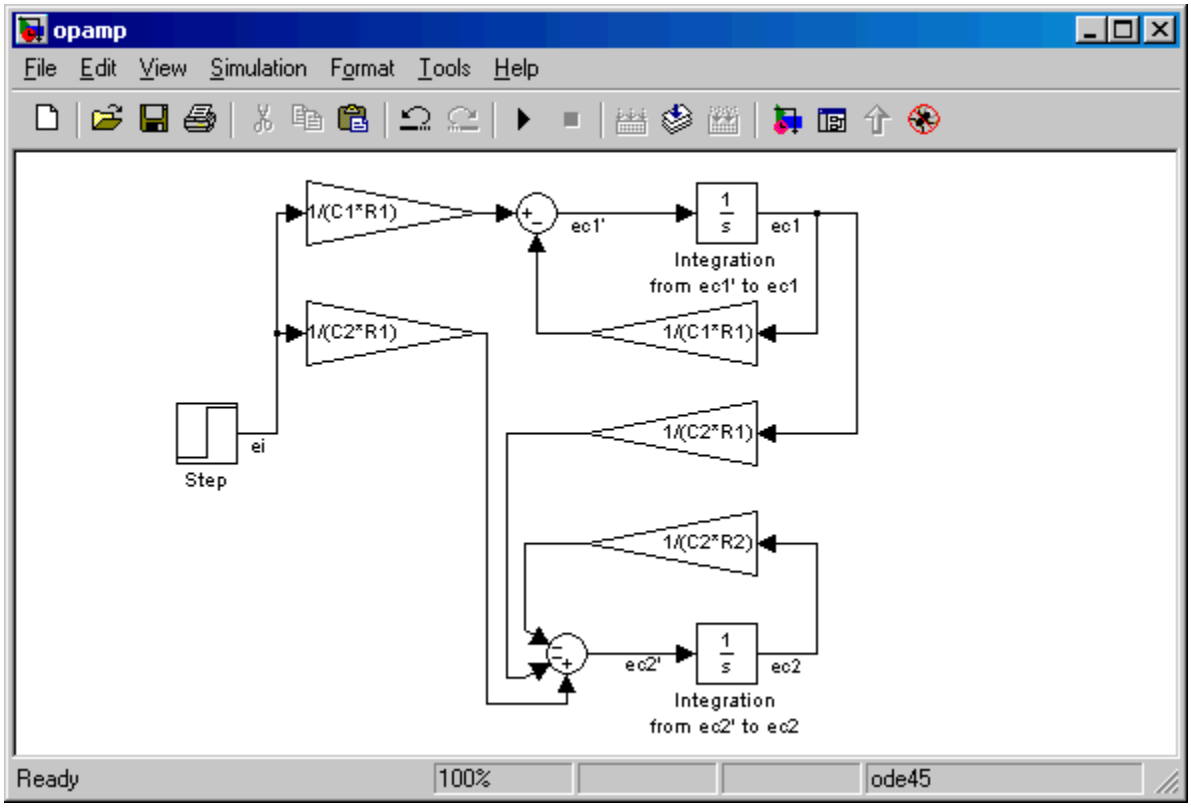
- a gain of 1/(C1*R1)
- a gain of 1/(C2*R1)
- a gain of 1/(C2*R2)
- a gain of 1/(C2*R1)

Add a Sum block as follows:

- Simulink Library Browser → Math Operations → Sum block
- change one of the inputs to a negative input by
    - double-click on the Sum block
    - change the List of signs from |++ to |+-

Similarly add a sum block of list of signs |--+

Connect wires as shown below

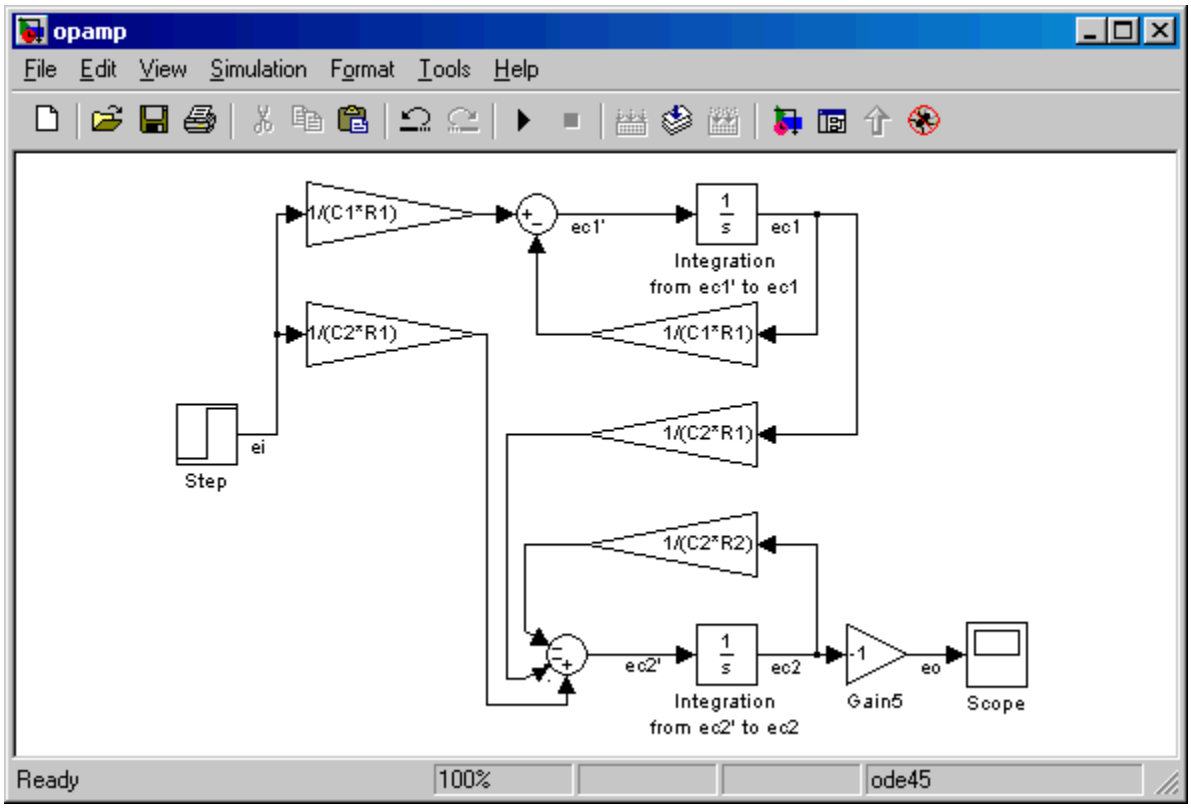


Add a Gain block with value -1 to the system.

To add a scope to monitor the output as follows

- **Simulink Library Browser** → click on Sinks→ drag the Scope sink 

Connect the two blocks as shown below:

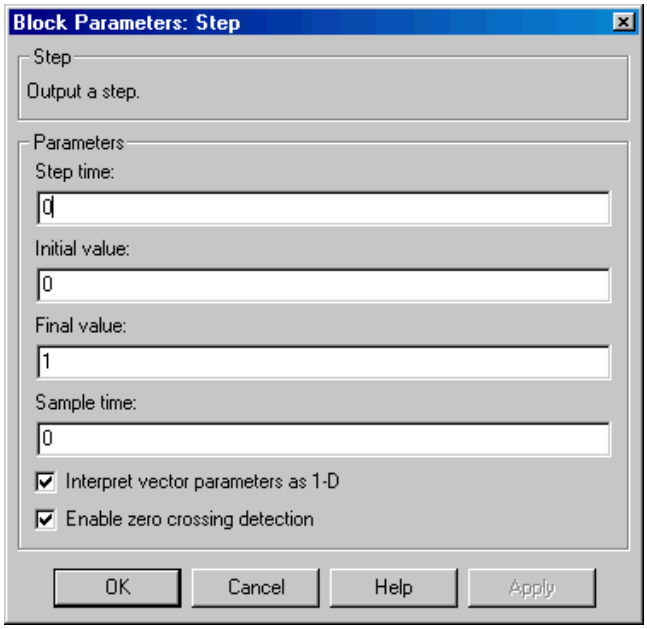


Save the model by selecting File in the toolbar of the model and selecting Save
The model is now complete and ready for running.

Enter the following values into the MATLAB window prompt.

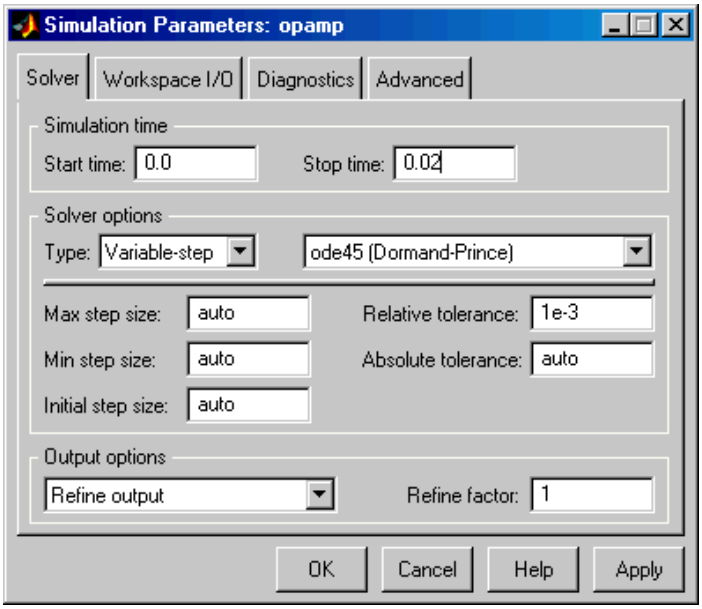
- R1 = 20000; R2 = 500; C1 = 0.1E-6; C2 = 0.1E-6;
- Press enter

To set the parameters of the step input, double click on the Step source and change the Step time to 0.

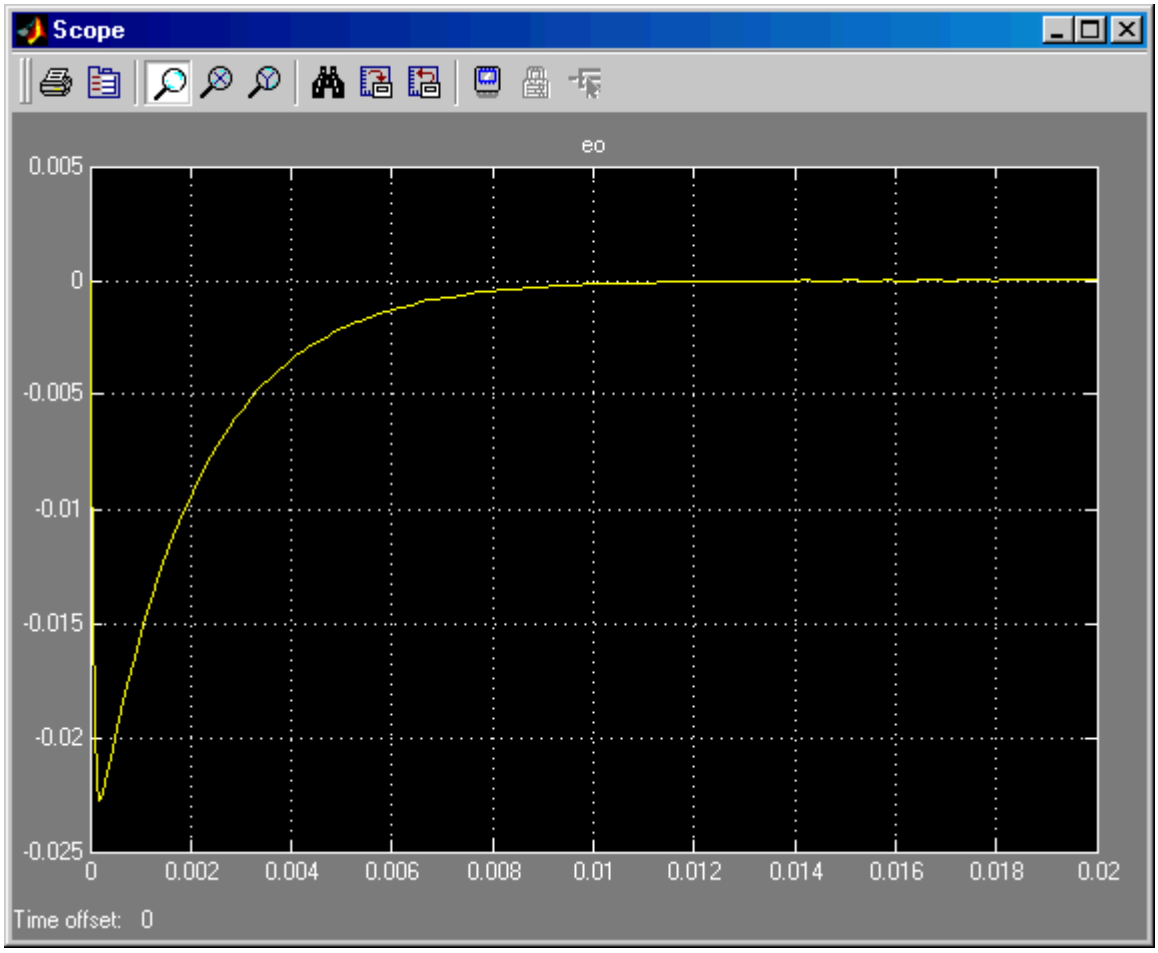


The default simulation time is from 0.0 seconds to 10.0 seconds. From previous analyses of this system we know that it reaches steady state within 0.02 seconds. To change the Simulation time,

- click on Simulation on the model toolbar
- select Simulation parameters
- change the Stop time to 0.02



Click the button on the tool bar to run the simulation.

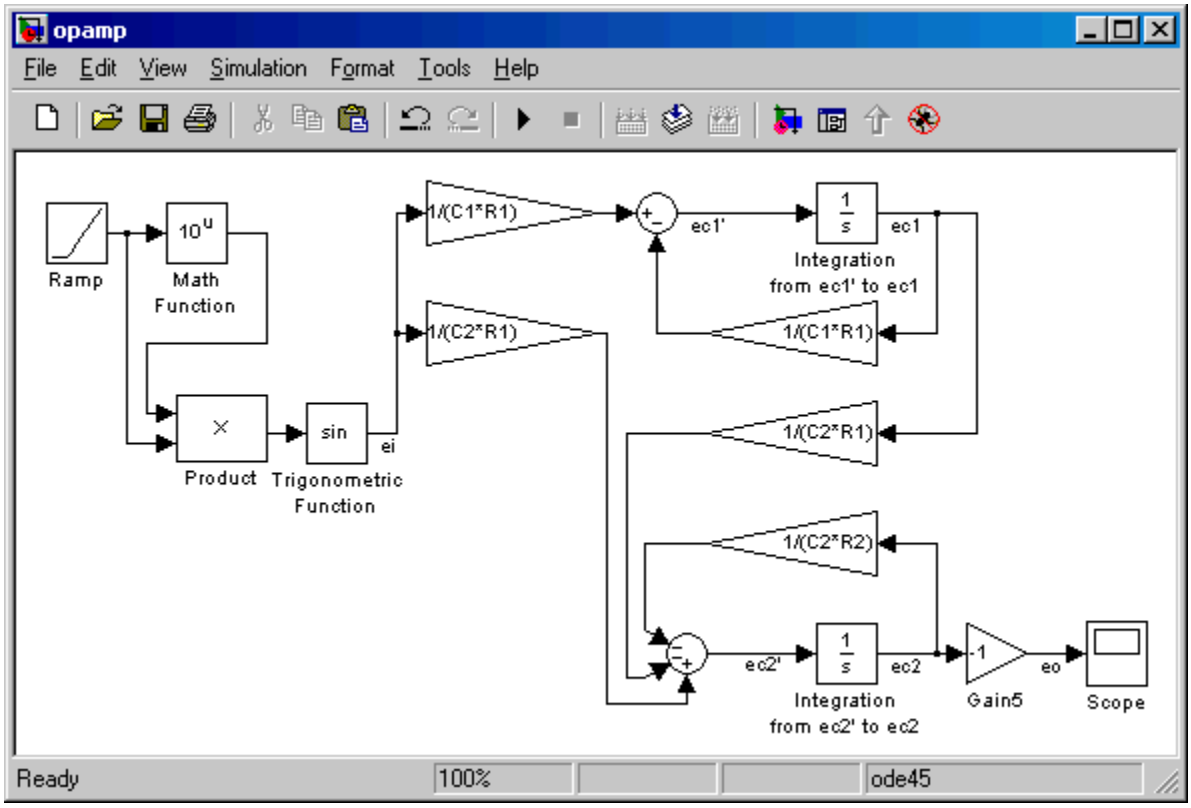


FREQUENCY SWEEP

- Delete the step input by

- Simulink Library Browser → Sources → drag the Ramp source 
    - Set the slope of the Ramp source to 1
- Simulink Library Browser → Sources → drag a Math Function block 
    - Set the math function to 10^u,
- Simulink Library Browser → Math Operations → Product block 
- Simulink Library Browser → Math Operations → Trigonometric Function 
    - Set the trigonometric function to sin.

Connect the components as shown below:



Set the simulation time to run from 0 to 5 seconds,