The simulation of any linear constant coefficient differential equation (LCCDE) can be represented by a block diagram. First rearrange the differential equation with the highest derivative on the left hand side of the equation. Represent this equation with a summing junction. For example, the equation
\[ \frac{d^2 y}{dt^2} + 5 \frac{dy}{dt} + 30y = 10u \]
is rearranged to:
\[ \frac{d^2 y}{dt^2} = -5 \frac{dy}{dt} - 30y + 10u \]
and represented by:

Next form a cascade of integrators to yield the lower order derivatives.

Finally, feedback the integrator outputs to the summing block and insert gain blocks.
coefficient *difference* equations, replacing the integrator with the unit delay block.

For example, the difference equation

\[ y[n] - 3y[n - 1] + 2y[n - 2] = u[n] + u[n - 1] \]

is simulated with the block diagram

```
1
z
```

**Simulink**

Simulink, which runs under MATLAB allows you to simulate dynamic systems in the time domain using block diagrams.

Simulink is introduced by means of an example of simulating the step response of a causal LTI system described by a second order differential equation:

\[ \frac{d^2 y}{dt^2} + \frac{dy}{dt} + 4y = 8u \]

We shall work through the following steps for simulating the system:

A  Build the block diagram
B  Choose the simulation parameters
C  Run the model
D  Save the model
A  Build the block diagram

• Start MATLAB
• Click on the Simulink icon on the MATLAB toolbar or type >>simulink

Starting Simulink displays the Simulink Library Browser

The Library Browser displays a tree-structured view of the Simulink block libraries. You build models by copying blocks from the library into a model window.

• Click the New button on the toolbar. This opens a blank model window on which you will place your blocks.
• Move the model window so that you can see the Library Browser
• Select Simulink => Continuous from the Library Browser
• Drag the Integrator icon to the model window. When it opens it will show an Integrator block with input and output ports.
• Drag a second Integrator block to the model window.
• Select Simulink => Math Operations. Drag 3 Gain blocks and 2 Sum blocks to the model window.
• Select Simulink => Sinks. Drag a Scope block to the model window.
• Select Simulink => Sources. Drag a Step block to the model window.
You now have all the blocks for the model. Arrange your blocks in the window to look like this:
By default, signals flow through a block from left to right. You can change the orientation of a block by choosing the **Flip Block** or the **Rotate Block** commands from the **Format** menu in the model window.

- Flip the two **Gain** blocks Gain1 and Gain2 to point from right to left.
- Rotate the lower **Sum** block so that the output port points up.
- Select the **Gain** blocks and change their values to 8, 1 and 4.
- Select the upper **Sum** block and change the signs to +-
- Select the **Integrator** blocks and check that the initial conditions are set to 0
- Select the **Step** block and set the step time to 0

To connect the blocks together, click over the output port of the first block and drag the pointer to the input port of the second block. To branch from an existing connection line, click the right mouse button over the line and drag the pointer to the input port of the target block.

- Connect the blocks together to produce the block diagram shown below:

**B Choose the simulation parameters**
Simulink solves differential equations by putting them into finite difference form and stepping in time through the simulation until a stop time is reached. The algorithm for solving the equations can be chosen by the user as well as the step size and other parameters. You set the simulation parameters and select the solver by choosing **Configuration Parameters** from the **Simulation** menu in the model window. Simulink displays the Configuration Parameters dialog box, which uses several pages to manage the simulation. The two key pages are:

1. The **Solver** page, which allows you to set the start and stop times, to choose the solver and to specify numerical step times and solution tolerance limits.
2. The **Data Import/Export** page, which manages input from and output to the MATLAB workspace from which you started.

There are also **Diagnostics** pages which allow you to select the level of warning messages displayed if you make a mistake (which of course never happens). For this first run, you will accept all the default values that Simulink has chosen in its wisdom. So you may proceed directly to the next step.

**C  Run the model**

- Open the **Scope** block.
- Select **Start** from the **Simulation** menu in the model window.

Your computer beeps to signal the completion of the simulation. The scope block should display the result …. a typical step response of an underdamped system.

**D  Save the model**

- Save your model in your own directory as **example1.mdl**. Simulink generates a specially formatted model file with the .mdl extension that contains all the properties of the model.

The results of the simulation can also be made available as vectors of response values that you can access from the MATLAB command window.

- Add an output to your model from the Simulink Library Browser. **Simulink** => **Ports & Subsystems** => **Out1**
• Select Configuration Parameters from the Simulation menu and choose Data Import/Export. Make sure that the time and output boxes in the “Save to Workspace” area are checked.
• Run the simulation again.
• Return to the MATLAB command window and list all the current variables in the workspace:
  >> whos
  You should find both a vector of time valuestout and a vector of output valuesyout listed
  >> plot(tout,yout)

That completes the example. You should now be able to build and simulate your own block diagrams in Simulink. Before we finish however, here are some tips for running your Simulink models:

**Increasing Resolution**

You may want to look at a specific part of the response and increase the number of simulation points
• Choose Configuration Parameters=>Solver=>Stop time =>3.
• Choose Configuration Parameters=>Data Import/Export=>Refine Factor =>4.
This gives you four times as many points over a smaller time range.
• Run the simulation again
• Return to the MATLAB workspace, plot the results and use the plot editor to find the maximum overshoot (2.8887)
Creating Subsystems

As you add additional blocks for more complex systems, the model can become cluttered. You can simplify your block diagram and make it easier for yourself and others to read by grouping blocks into a new subsystem:

- Enclose all the blocks in example1.mdl except for the Step, Scope and Out1 blocks within a bounding box. (Click on empty space and drag to form a box)
- Choose Create Subsystem from the Edit menu.

After resizing and rearranging for aesthetic delight, your model should look like:

If you open the subsystem block, the underlying system is shown beneath. Simulink adds Inport and Outport blocks to represent the inputs and outputs to the subsystem.

Now follow the steps to simulate the step response of a discrete system given by the linear constant coefficient difference equation:

\[ y[n] - \frac{1}{4} y[n-1] + \frac{9}{16} y[n-2] = x[n] \]

A Build the model

- Set Sample time = 1 in the step input block

B Choose parameters
Set solver options to discrete. Set start time to 0, stop time to 20.

C Run model

Use the command `>>stem(tout,yout)` for discrete time plots

D Save the model

Save as `example2.mdl`

Congratulations. You are now a Simulink Super User!