

Arcade Tutorial 2: Elastic Beam

The following tutorial demonstrates the following Arcade features beyond tutorial 1:

- Adding loads.
- Controlling loads over time.
- Using elastic beam elements.
- Drawing moment diagrams.
- Getting detailed information about element forces.

Build the Model:

- **Turn on the snap grid.** (*Settings > Graphic > Grid*, upper window)
- **Add two nodes as shown below:** (*Build > Nodes*)



- **Add a Beam-1 Element**
 - Click *Build > Elements > Beam-1*.

The Beam-1 element is an elastic beam element. Like the truss element, it resists the nodes getting closer together or farther apart. It also bends and flexes, resisting changes in relative rotation between the end nodes.
 - In the left window, set the following properties:
 - *Section Area: 20*
 - *Moment of Inertia: 500*
 - *Material: steel*
 - *Depth: 30*

The depth is used only for certain rendering options. It is not used as a physical property of the element.
- **Add damping**
 - Click *Settings > Damping > Beam-1*.
 - In the *Beam-1 mass-proportional damping* area, click the button labeled *Heavy*.

- **Subdivide the element**
 - Click *Build > Subdivide Element*.
 - In the left window, use the '>' button to set the number of subdivisions to 4.
 - Click on the beam element in the Build View.

The element is subdivided into four equal-length elements.

- **Add supports**
 - Click *Build > Supports*
 - Click on the leftmost node.

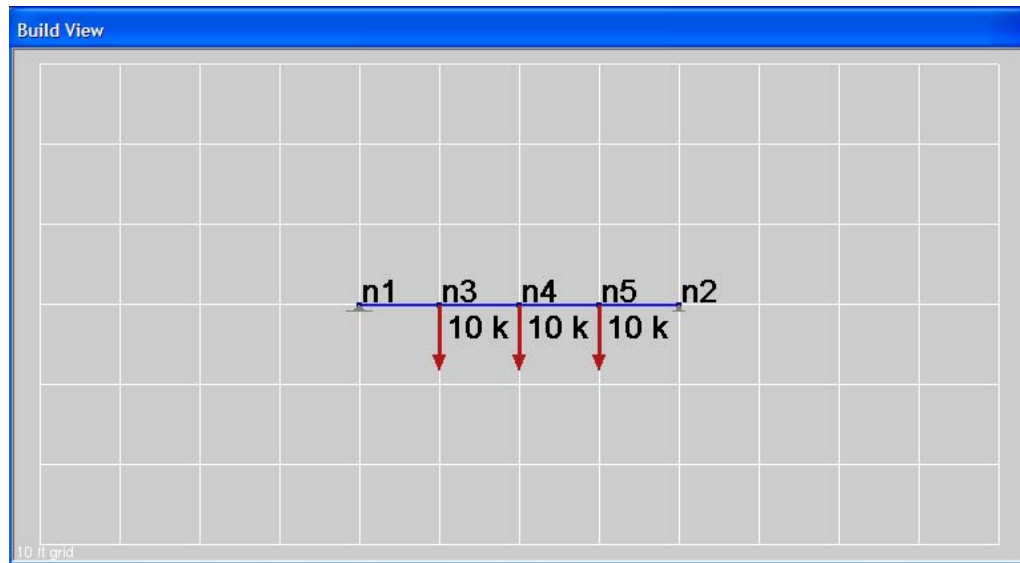
A pin support is added.
 - In the left window, click on the button labeled *Horiz. Roller*
 - Move to the Build View and click on the rightmost node.

A horizontal roller support is added.

- **Add loads**
 - Click *Build > Loads*.
 - In the left window, set the following:
 - Physical properties
 - *X*: 0
 - *Y*: -10
 - *Moment*: 0
 - Graphic properties
 - *Draw tail at node*
 - Load pattern
 - *pat1*

All loads belong to a load pattern. Loads belonging to a pattern can be scaled up or down as a group over time, using methods described below. The default pattern, initially named *pat1*, specifies the loads to ramp from a scale of 0 to 1 during the first second of simulation time.

- In the **Build View**, click on the three interior nodes of the beam.
The model should appear similar to below.



Note:

Click *Settings > Graphic > Forces*, and in the upper window select *Bold* under *Arrow Style* to display the forces as shown in the image.

Simulate

- **Run the simulation**

- Click *Simulation > Start*.

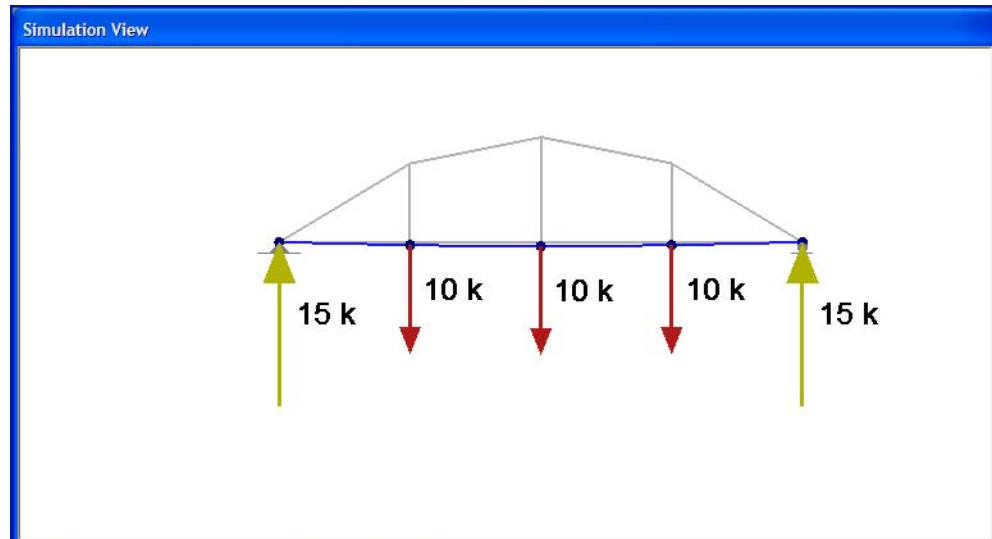
The simulation starts, and the loads ramp from zero to their full value during the first second of simulation.

- **Show reactions and moment diagram**

- Click *Settings > Graphic > Supports*.
- In the *Reaction forces* group, click *Display* and *Magnitudes*.
- Click *Settings > Graphic > Elements*.
- In the *Moment diagram* group, click *Display* and *Draw on compression side*.

- Use the buttons to adjust the graphic scale until the moment diagram looks right to you.

The model should appear similar to below, after adjusting the view using *View > Zoom by rectangle*.



- **Find the magnitude of the bending moment**

- Click *General > Show info*.
- In the left window, in the *Element info group*, click *Beam-1* from the dropdown list.
- Click the *Moment i* and *Moment j* checkboxes.
- Move to the Simulation View, and click on the second element from the left.

A popup window appears with the following information.

Popup window information	Meaning
Beam-1: e3	The element type is <i>Beam-1</i> . The element tag is <i>e3</i> .
M n3: -150 k-ft	The start end (i-end) of the element is at node <i>n3</i> , the moment at that end is -150 k-ft. (counterclockwise positive).
M n4: 200 k-ft	The finish end (j-end) of the element is at node <i>n4</i> , the moment at that end is 200 k-ft.

Modify the Load Control

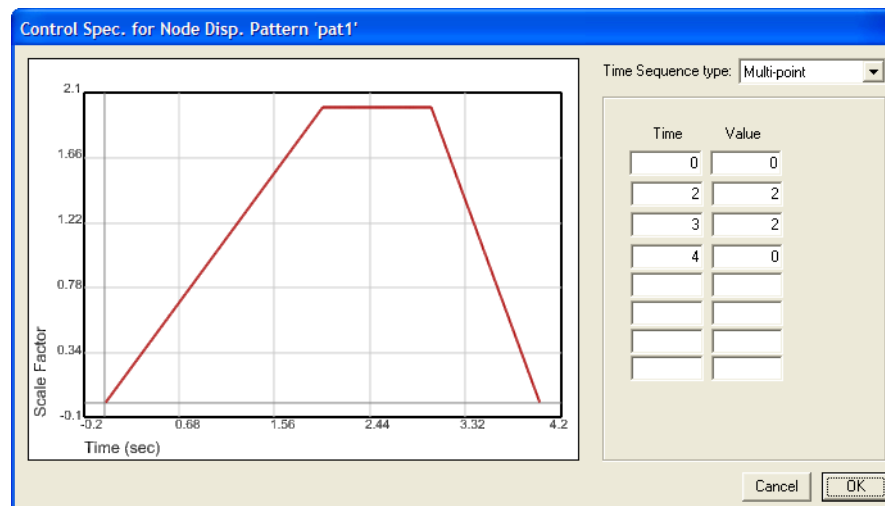
The model used the default load control, which is to ramp loads from zero to full value during the first second of simulation. The following steps demonstrate how to change the ramping of loads, plus other means of defining load control.

- **Change the multi-point specification**

- Click *Simulation > Stop* to end the simulation.
- Click *Build > Loads*, and then click on the *Load patterns* tab of the table set in the upper window.
- In the *Load patterns* table, click on the words *Multi-point...* in the *Control spec.* column.

A dialog box appears, showing the data and graph for the Load Control Specification. A Multi-point specification is defined by a series of time-value pairs, where the values represent a scale factor applied to all loads in the pattern. The scale factor ramps linearly from point to point.

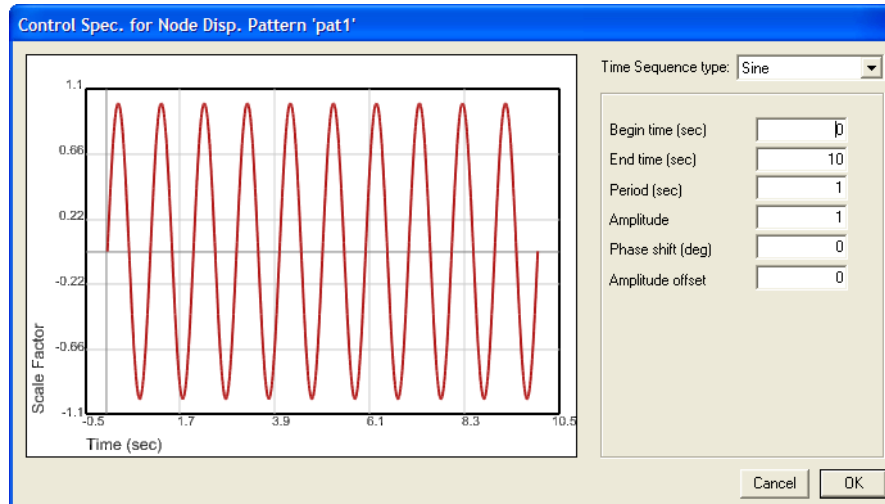
In the dialog box, insert time value pairs as shown in the figure below.



This specification ramps the pattern from 0 to 2 during the first 2 seconds of simulation, holds that value for one second, and then ramps down to 0 over one second.

- Click OK in the dialog box.
- Click *Simulation > Start* to run the simulation again.
The loads follow the new specification, each ramping up to 20 kips over two seconds, holding for a second, and then ramping down to zero.
- Click *Simulation > Stop* to end the simulation.

- **Use a sine specification**
 - In the *Load Patterns* table, click on the words *Multi-point...* in the *control spec.* column.
A dialog box appears.
 - From the dropdown list labeled *Time sequence type*, select *Sine*.
The dialog box appears as shown below:



The dialog now lets you set properties to control the load pattern using a sine wave. The value of the sine curve at any time defines the value of the scale factor for the load pattern.

- Click OK.
- Click *Simulation > Start* to run the simulation again.
The loads follow the new specification, Oscillating at a period of one second for ten seconds.

Additional Modifications

Here are suggestions for further modifications to the model.

- Add a hinge release in the middle of the beam. Click *Build > Elements > Beam-1*. In the *Beam-1* table, click on the *i-rel* property for element *e3* or the *j-rel* property for element *e4* and set it to *moment*. The *i-rel* property puts a hinge (moment release) at the i-end (start end) of an element, and *j-rel* puts it at the j-end. Run the simulation and see what happens.
- Run a simulation and try the shake tool. Click *Simulation > Shake model*, and then click and drag in the simulation view.