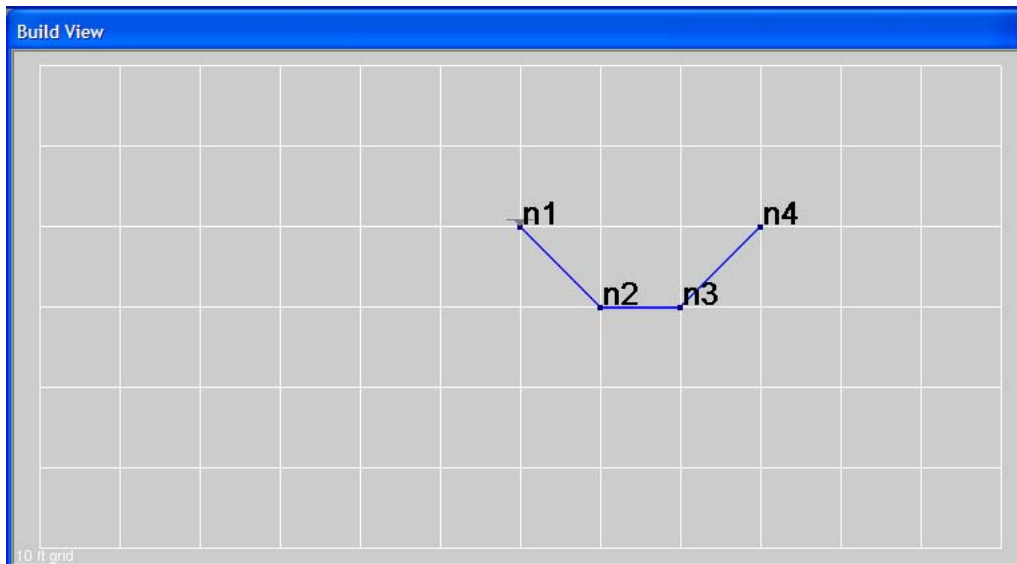


Arcade Tutorial 1: Complex Pendulum

The following tutorial demonstrates basic model building and simulation in Arcade, including

- Adding nodes.
- Using elastic truss elements.
- Using supports.
- Modifying physical and graphic properties.
- Creating graphs of model data changing over time.

The subject of the tutorial is a complex pendulum.



Build the Model:

- **Turn on the snap grid.** Click *Settings > Graphic > Grid*. In the upper window, check the box labeled *Snap grid on*.
- **Add Nodes.**
 - Click *Build > Nodes*.
 - Move the mouse to *Build View* and click nodes at four points as shown above.
- **Add a Truss-1 Element**
 - Click *Build > Elements > Truss-1*.
The Truss-1 element is an elastic truss element. It acts as a spring that resists the nodes getting closer together or further apart.
 - Click from node *n1* to *n2*, to *n3*, to *n4* as shown.
- **Add a Support**
 - Click *Build > Supports*
 - Click on the leftmost node.

Note:

Click *Settings > Fonts* and then *All Medium* to change the font display to match the tutorial image.

Simulate:

- **Start the simulation**
 - Click *Simulation > Start*
The background turns white, indicating the simulation has started, but nothing is moving yet.
- **Turn on gravity**
 - Click *Settings > Gravity*.
 - Next to the *Accel. of gravity* property, click the button labeled *Ig*.
The pendulum starts swinging as gravity is applied. Note the *Ramp gravity* option. When this is checked, the acceleration of gravity is ramped up from zero over the time specified by the time below the checkbox. Ramping gravity reduces the dynamic shock to a model that occurs when gravity is applied instantly.
- **Stop the Simulation**
 - Click *Simulation > Stop*.

Modifying the model

You can modify the model and its display as follows:

- Modify physical properties of the model using tables, which are displayed with each Build Tool (e.g. *Nodes*, *Supports*, etc.).
- Modify the graphic rendering of the model using the Graphic Settings windows. Click *Settings > Graphic* to access these.
- Modify the graphic display of individual supports and loads by right-clicking on them.

Modify Physical Properties of Nodes and Elements

- **Modify Node Coordinates.**
 - Click *Build > Nodes*.
 - In the *Nodes* table of the *Nodes Table Set*, change the Y coordinate of some of the nodes.
- **Modify Truss-1 Properties**
 - **Add a new material and assign it to the element.**
 - Click *Build > Elements > Truss-1*
 - In the *Truss-1 Elements Table Set*, click the *Truss-1 Materials* tab.
 - Click anywhere in the first row, and hit the Enter key.
A new row will appear.
 - In the new row, type in the following values:
 - *tag*: alum
 - *E modulus*: 10000
 - *wt. density*: 0.15

These steps have created a new material with the properties of aluminum.

- Click the *Truss-1 Elements* tab in the table set.
- For one of the elements, click on the word *steel* in the *material* column. (A popup menu appears).
- Select *alum* from the popup menu.

This step assigns the new material to the element.

Modify Graphic Properties

- **Start the Simulation**
 - Click *Simulation > Start*.
- **Display reactions**
 - Click *Settings > Graphic > Forces*.
 - Click the checkboxes *Reactions* and *Magnitudes in Sim. View*.
 - In the *Graphic Scales* section of the *Graphic Settings* window, use the adjust buttons (<<. <. >. and >>) to set the *Force* scale factor so that the reactions are drawn at a reasonable size.
- **Modify the display of the support**
 - Right click on the symbol for the support at the top of the pendulum. (a dialog box appears).
 - For the *Draw Support*, select *above*.
 - For the *Draw Horiz. Reaction* option, select *tail at node*.
 - For the *Draw Vert. Reaction* option, select *above*.

Modify Units of Measurement

- Click *Settings > Units*.
- In the *Unit Settings* window, for *Force* select *lb*.

The reaction magnitudes will be displayed in pounds rather than kips. Changing units of measurement affects only the way that values are reported, it does not change the physical properties of the model.

Add a Graph

Graphs provide a means to examine how the state of the model changes with time. The following steps will add two graphs, one that plots the X displacement of the pendulum tip with respect to time, and another that plots the Y displacement with respect to the X displacement.

- **Add a node monitor**
 - Click *Build > Node Monitor*.
 - In the left window, set the *End Time* to 30 seconds, and the *Quantity* to *X disp*.
 - In the Build view, click on the node at the tip of the pendulum.

During a simulation, a node monitor records the selected quantity during the interval from the Begin Time to the End Time. The data from the monitor is used to create graphs.

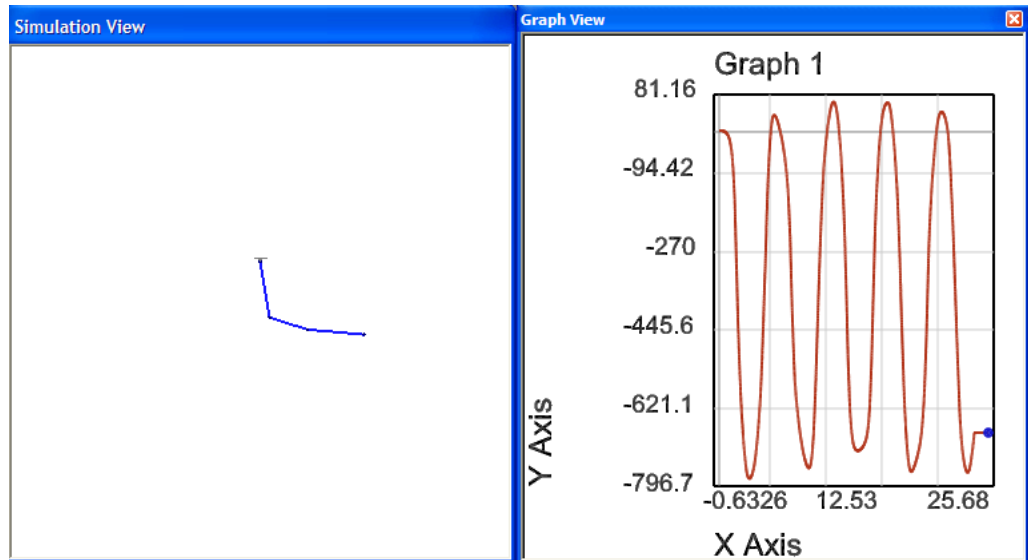
- **Add a graph and view it**

- Click *Graphing > Add Graph*.

A new graph window appears adjacent the Build View, along with a left window for defining what information to graph, and an upper window defining how to graph it.

- Click *Simulation > Start* to run the simulation.

The graph will begin plotting the monitored X displacement vs. Time. After 30 seconds, the Build View and Graph View should look like the following:



The graph flatlines at 30 seconds because the node monitor has stopped recording.

- Click *Simulation > Stop* to stop the simulation.
- Right click anywhere on the plot line, a popup menu with detailed information appears.

Left click on the popup to dismiss it.

- **Add another node monitor, and create another graph**

- Click *Build > Node Monitor* to add another node monitor.
- In the left window, set *Quantity* to *Y disp.* and click on the node at the tip of the pendulum.
- Click *Graphing > Add Graph*.
- In the left window, in the *X Axis* group, set the first value to the X displacement of the node.

The choice will read something like "node n4 disp. X". The node number may be different depending on the order you clicked the nodes in the beginning.

- In the same window, set the first value of the *Y Axis* group to the Y displacement of the node.
- Click *Simulation > Start* to run the simulation.

The graph will begin plotting the monitored Y displacement vs. the monitored X displacement, effectively tracing the trajectory of the pendulum tip.

Note:

The upper *Graph Settings* window includes several options to change the graph display and add descriptive labels. Try changing the options to see the effect on the graph.

Additional Modifications

Here are suggestions for further modifications to the model.

- Try different levels of damping on the nodes (see *Settings > Damping > Nodes*).
- Try using the Hook Tool (*Simulation > Hook a node*) to pull on a node. In order to click on a node you may want to pause the model (*Simulation > Pause*, or F7), click on the node, and then resume the simulation (*Simulation > Pause*, or F7, again). You can then click and drag to pull on the node.