Arcade Tutorial 3: Braced Frame with Earthquake

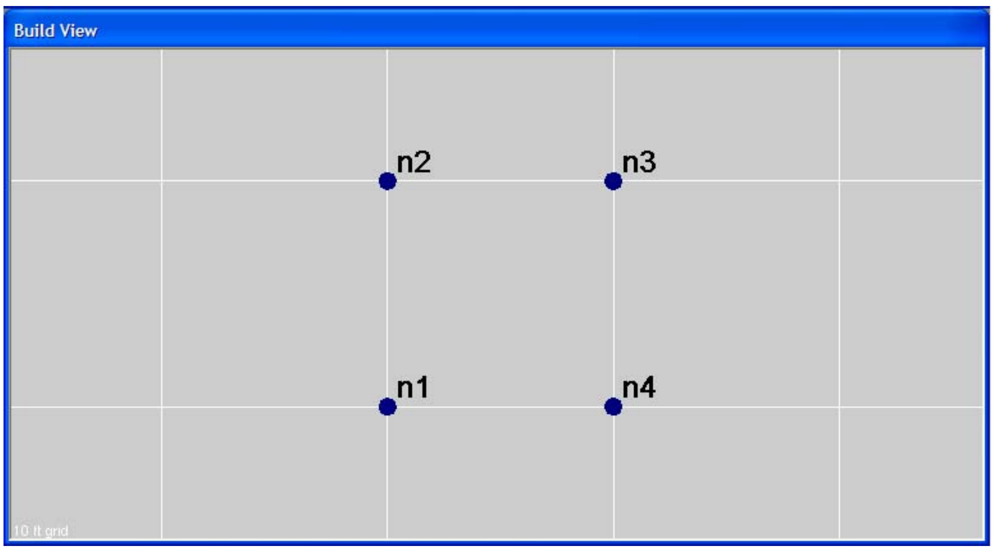
The following tutorial demonstrates the following Arcade features beyond preceding tutorials:

- Adding seismic ground motions as nodal acceleration patterns.
- Using inelastic truss elements.
- Using graphs that combine data from node monitors.

Build the Model:

- **Turn on the snap grid**. (*Settings > Graphic > Grid*, upper window)
- **Add four nodes as shown below, and zoom the view appropriately:** (*Build > Nodes*) (*View > Zoom by rectangle*)
  - After adding the nodes, go to the node table and set the *super. wt.* of nodes *n2* and *n3* to 40.

    These values represent superimposed weight in addition to the weight of the frame itself.



- **Add columns and girder as Truss-2 elements.**
  - Click *Build > Elements > Truss-2*.

    The Truss-2 element is an inelastic truss element. It models tension and compression yielding, as well as tension fracture. The yield stress and fracture strain are defined in the *Truss-2 Materials* table.
  - In the left window, set the following properties:
    - *Section Area*: 10
    - *Material*: steel
  - Click from node *n1* to *n2*, to *n3*, to *n4*.

Kirk Martini © 2005
University of Virginia

- **Add tension-only braces**.
  - In the left window, change the properties to the following:
    - *Section Area*: 1
    - *Material*: steel
  - Click from *n1* to *n3*, then right click in the background.
  - Click from *n2* to *n4*.
  - Click *Build* > *Subdivide element.*
  - In the left window, use the buttons to set *Number of subdivisions* to 3.
  - Click once one each of the diagonal elements.

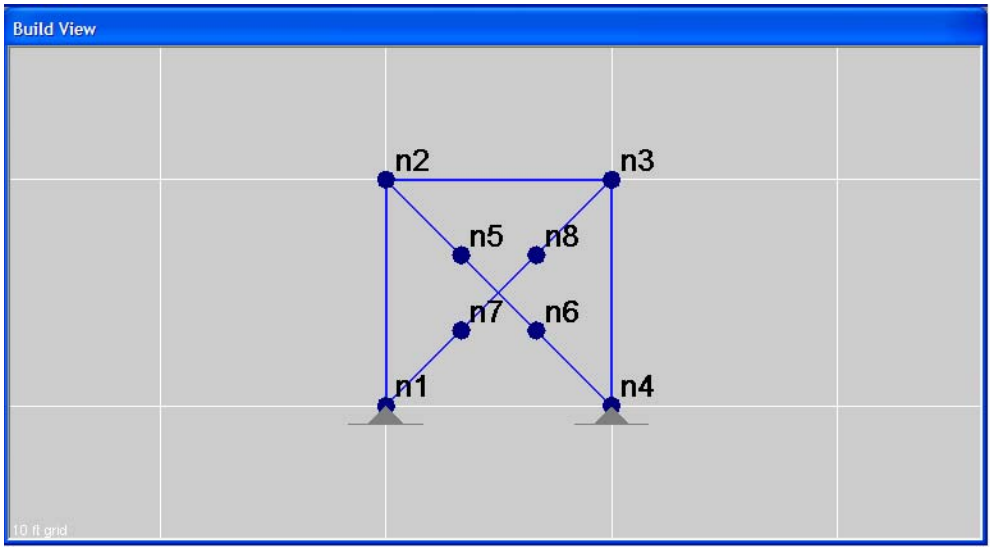
- **Add damping:**
  - Click *Settings* > *Damping* > *Truss-2.*
  - In the upper window, in the area labeled *Truss-2 mass-proportional damping*, click the button labeled *Heavy*.

    Clicking the *Heavy* button sets the damping to a value that produces large damping for many medium-scale structures, but the behavior can vary considerably depending on the specifics of the model. This particular model actually needs more damping than *Heavy.*
  - Then, click the button labeled '>' two times so that the value of damping rises to 20,000 (2e+4).

    This model needs significant element damping because of the vibrations of the cross braces when they snap into tension.

- **Add supports**
  - Click *Build* > *Supports*
  - Click on the bottom two nodes.

    The model should look similar to the following:



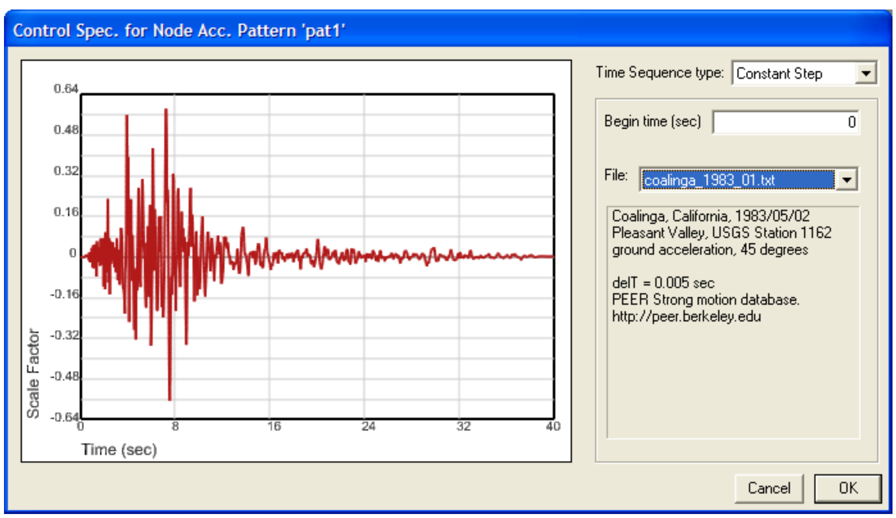
- **Add earthquake ground motion as nodal acceleration pattern**
    - Click *Build > Node accelerations.*
    - In the left window, leave the default values unchanged. These values are
        - *Direction*: X
        - *magnitude*: 1
        - *Node acc. pattern*: pat1

          These values correspond to a nodal acceleration of 1 g in the positive X direction. The variation of the acceleration over time will be defined by a control specification associated with the pattern, in the same way that the variation of loads over time was defined in tutorial 2.
        - Click on nodes n1 and n4.

          The accelerations for these nodes are added to the *Node Accelerations* table.
        - Click on the *Node Acceleration Patterns* tab of the table set.
        - 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 *Constant Step*.

          The dialog box appears as shown below:



          A constant step control specification defines variation over time by reading data from a file. The first value in the file is a time interval in seconds, and the remaining values define the variation at steps by that interval. It is well suited to seismic records where there are many time points.
        - From the dropdown list labeled *File*, select *coalinga_1983_01.txt.*

          The graph in the dialog box shows the acceleration record for this ground motion.
        - Click OK.

Simulate:

- Click *Simulation* > *Start.*

  The model shakes in response to the earthquake. The diagonal members go slack because yielding causes permanent stretching.

Add a Graph:

To more deeply understand the response of the structure to the earthquake, a graph can illustrate the relationship between the total lateral force on the structure (the *base shear*) and the lateral movement, also called *lateral drift*. The following steps describe how to add such a graph.

- **Add node monitors**
  - Click *Build* > *Node Monitors*
  - In the left window, set the following properties:
    - *Begin time*: 0
    - *End time*: 60
    - *Quantity*: disp. X
  - In the Build View, click on nodes *n1* and *n2* (the two left nodes).

    Data from these monitors will be used to graph lateral drift.
  - In the left window, change the *Quantity property* to *net force X.*
  - Click on nodes *n1* and *n4* (the two bottom nodes).

    Data from these monitors will be used to graph base shear.

- **Add and define graph**
  - Click *Graphing* > *Add Graph.*
  - In the left window, in the *X Axis* group, set the first dropdown list to *node n2 disp. X*.
  - Set the second dropdown list in that group to *node n1 X disp X.*
  - Set the Scale factor next to the second dropdown list to -1.

    You can do this by clicking the '+/-' button next to the scale factor.

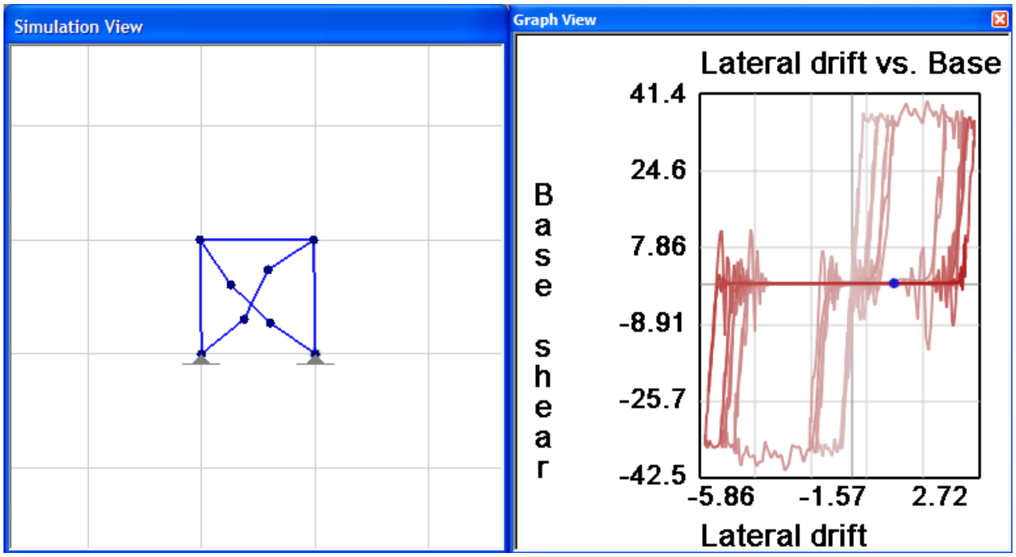
    With these steps, the X axis quantity for the graph is the X displacement of node *n2* minus the X displacement of node *n1*, in other words the lateral displacement of node *n2* relative to node *n1*. This is the lateral drift of this structure.
  - In the left window in the Y Axis group, set the first dropdown list to *node n1 net force X*
  - Set the second dropdown list in that group to *node n4 net force X*.

    With these steps, the Y axis quantity for the graph is the net X force at node *n1* plus the net X force of node *n4*, in other words the sum of the horizontal reactions. This is the base shear of this structure.
  - In the upper window, set the following properties:
    - *Title*: Lateral drift vs. Base shear
    - *X Axis*: *Label*: Lateral drift
    - *Y Axis: Label*: Base shear
    - Click the check box *Fade with time.*

Simulate:

- Click *Simulation* > *Start.*

  After about 30 seconds, the model and graph should look like the following:



  The graph illustrates the behavior of the structure in response to the earthquake. The diagonal braces yield during the early shaking, causing them to go slack. This slackness gives the structure a region of 'play', where it moves laterally with almost no resistance, corresponding to the flat part of the graph. The steep parts of the graph at either end of the flat region correspond to one of the braces tightening.

- To see the graph more fully, click *View* > *Toggle full view*.
- To see the graph rendered more smoothly, Click, *Graphing* > *Edit Graph.*
  - In the upper window, set *Sample interval* to 0.005.

Additional Modifications

Here are suggestions for further modifications to the model.

- Try other earthquake motions. Stop the simulation if it is running. Click *Build* > *Node accelerations*, and click the *Node Acceleration Patterns* tab in the table set. Click on the cell that says *Const. step* and select another motion from the dialog box.

  The files available from the Constant step dialog are kept in the folder "c:\Program Files\Arcade\Arcade\const_step_files". You can put your own files there, following the format that the first number is the time step, and the remaining numbers specify values at that constant step.

- Turn on gravity. So far, the simulation has not included the effects of gravity. Click *Settings* > *Gravity*. In the upper window, click the button labeled *1g*. Run the simulation again. Note the resulting changes in the graph.