ETABS 2016 Tutorial: Frames

Below is a tutorial that was organized for educational purposes at Christian Brothers University only. The procedure of analysis in ETABS 2016 is similar to that of ETABS v9.

Example

Draw the shear force and bending moment diagrams for the frame below. The connection at C is rigid. Both members are made from the same material and have the same cross-section properties.

Solution

Step One: Open ETABS.

Step Two: Select “New Model”.
Step Three: Select the Initialization Options. For this example, we will select “Use Built-in Settings”.
Step Four: Specify a grid spacing and story height (based on your problem). Note that the Z-coordinate defines the gravity direction. This is specified by the “Story Dimensions”. Working in the $xz$ plane, we will specify 2 grid lines in the $x$-direction, with a spacing of 10 ft and 2 stories, with a story height of 12 ft.

You may work in either the 2D or 3D window. Let us work in the 2D window for this example. In the 2D window, change the view to elevation 1. This will take us to the $xz$ elevation view.
Step Five: Define material properties.

Modify the material properties by making the mass and weight per unit volume zero. We will assume the material weight is zero so as to not induce unwanted shears and moments. Let’s modify the A992Fy50 Steel.
**Step Six:** Define frame sections.

For this example, we are given that both members have the same cross-section properties. Because of this, we can select any section, so long as we apply it to all members. Let us select a Wide Flange W10X12 member for our two members. By clicking on “Modify/Show Property”, we can see the default properties of the W10X12.
**Step Seven:** Now we shall draw our members.

Make sure you select the W10X12 in the “Properties of Object” tab. Change the existing hinge at the left, bottom support to a roller by selecting “Assign”, then “Joint”, then “Restraints”. Also, select the right external support and change it to an external hinge.
The default for ETABS is a frame. That is, when you draw a structure in ETABS, it automatically assumes a frame structure. Since this is an example of a frame, we should not “release moments” as we would do for a truss. We can also display an extruded view of the frame. This shows the actual shape of the members. To show this, click the check box and select “Extrude Frames”. Below is a 3D figure of our extruded frame.
Set View Options

General

View by Colors of: Objects

Objects Present in View
- Joint Objects
  - Invisible
- Columns
- Beams
- Braces
- All Null Frames
- Floors
- Walls
- Openings
- All Null Shells
- Wall Stacks
- Links
- Tendon
- Design Strip Layer A
- Design Strip Layer B
- Design Strip Layer Other

Special Effects
- Object Shrink
- Object Fill
- Object Edge
- Extrude Frames
- Extrude Shells

Other Special Items
- Joint Restraints and Springs
- Daphragm Extent
- Connections
- Story Labels
- Dimension Lines
- Architectural Plan Layers
- Horizon
- Shell Analysis Mesh
- Slab Internal Ribs
- Isolated Column Footings
- Soil Profile for Joints
- Soil Profile for Areas

Apply to All Windows

Set to Default View Options

OK  Close  Apply
**Step Eight:** Next, we shall apply our loads. Select the members where the loads are to be applied. Under the “Assign” menu, select “Frame Loads”. For member CD, we will assign a uniform distributed load of 0.1 klf in the Gravity direction. For member AC, we will assign a point load at a relative distance of $5/12 = 0.4167$ from A in the Global-X direction. For now, we will only consider one load “combination” and we will consider it to be a dead load.
Step Nine: Now we run our model by selecting the “play” button. The model will then show the deformed shape of the frame.

To display the shear force and bending moment diagrams, select the button shown below. The shear force diagram is given under “Shear 2-2” and the bending moment diagram is given under “Moment 3-3”.

![Diagram showing the deformed shape of the frame and shear force and bending moment diagrams.](image)
Below are the shear force and bending moment diagrams, respectively.
If you noticed that something seems off about these diagrams, you are correct. The shear diagram should not have a linear segment from the roller to the 5 ft location on AC. To eliminate this misleading output, “unlock” the model, select both members, click “Assign”, “Frame”, and “Output Stations”. Change the “Min Number Stations” to a large number. Let’s set it to 5,000. What this means is that ETABS will report and plot internal loads at 5,000 evenly-spaced locations along the two frame members. Also, we can change the “End Length Offsets” to zero in order to display the diagrams over their appropriate lengths.

Re-running gives the correct shear and moment diagrams, respectively.
We can also determine the displacement of a joint by right clicking on a desired joint.