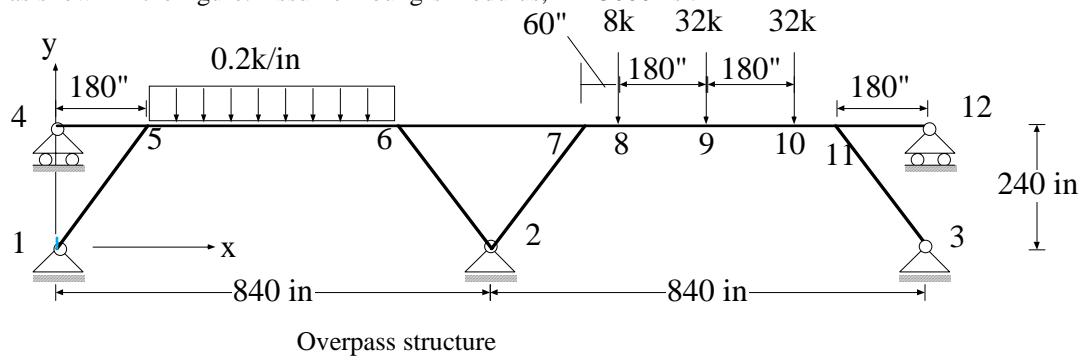


Ansys Lab — Frame Analysis

Analyze the highway overpass frame shown in Figure. The main horizontal beam is W24x162 (area = 47.7 in², moment of inertia = 5170 in⁴, height = 25 in). The inclined members are W24x76 (area = 22.4 in², moment of inertia = 2100 in⁴, height = 23.92 in). The loading, given in kips (1 kip = 1000 lbs), is applied as shown in the figure. Assume Young's modulus, $E = 3000$ ksi.



Enter the ANSYS program by using the launcher

Enter a job name, say **Overpass > Run**

File > Change Title > [**2-D Beam**]

Set up the graphics area. The following settings work well.

- ANSYS Utility Menu
- Workplane > WP Settings > Grid and Triad
 - Snap Incr[**60.0**]
 - Spacing[**60.0**]
 - Minimum[**0**]
 - Maximum[**1680**]
 - > OK
- Workplane > Display Working Plane
- PlotCtrls > View Settings > Magnification > DVAL
 - User Specified [**1680**] > OK
- PlotCtrls > View Settings > Focus Point > XF
 - User Specified XF, YF, ZF [**840**] [**150**] [**0**] > OK

Preprocessing

1. Define the element type, element real constants, and material properties.

2D elastic beam is the appropriate element for analyzing plane frames.

- Preprocessor > Element Type > Add/Edit/Delete... > Add... > Structural **Beam** [**2D elastic 3**] > OK
- > Close

Since two different sections are used in the frame, we must create two different sets of real constants. The minimum real constant data for 2D beam elements consist of area of cross-section (AREA) and moment of

inertia (IZZ). Section height is used only during computation of stresses. It has no effect on displacements and bending moments

- Preprocessor > Real Constants > Add/Edit/Delete... > Add... > OK
Real Constant Set No. [1]: AREA [47.7], IZZ [5170], HEIGHT [25] > OK
- Real Constants > Add... > OK
Real Constant Set No. [2]: AREA [22.4], IZZ [2100], HEIGHT [23.92] > OK > Close

Both sections are made of steel. Therefore we need to define only one material. The only material property needed by 2D elastic beam is the Young's modulus.

- Preprocessor > Material Props > Material Models > Structural > Linear > Elastic > **Isotropic** > EX [3000] > OK
- ANSYS Toolbar > SAVE_DB

2. Create nodes by picking points on the working plane. List and plot the nodes

For frames consisting of uniform sections, it is sufficient to use one element between any two joints. The concentrated loads, however, can only be applied at joints. Since the right span of the overpass structure is subjected to several point loads it is necessary to divide this span into several elements, with nodes falling at locations of concentrated loads. The left span carries distributed load and thus is modeled by using only one element.

The model as described above gives exact nodal solutions for both displacements and moments. However solution within the element is exact only for those elements that are not subjected to distributed loads. Thus our present model will not give correct bending moment distribution for the left span. If you are interested in accurate solutions over elements with distributed loads you must divide the loaded span into several elements.

Ansys graphics has a limitation. For drawing deformed shapes the graphics system uses only the displacements and does not take into consideration the rotations. Thus if only one element per span is used then the deformed shapes involving beam elements do not look realistic. Once again if you are interested in seeing realistic deformed shapes you can divide each span into several elements.

- Preprocessor > -Modeling- **Create Nodes** > On Working Plane
[WP = (0, 0)]
[WP = (840, 0)]
[WP = (1680, 0)]
[WP = (0, 240)]
[WP = (180, 240)]
[WP = (660, 240)]
[WP = (1020, 240)]
[WP = (1080, 240)]
[WP = (1260, 240)]
[WP = (1440, 240)]
[WP = (1500, 240)]
[WP = (1680, 240)]
- PlotCtrls > Numbering > Plot Numbering Controls > NODE > **On** > OK
- PlotCtrls > Pan-Zoom-Rotate > **Fit** > Close
- ANSYS Toolbar > SAVE_DB

3. Define elements by picking nodes

When there are different real constants, we must select element attributes and choose appropriate real constant set before creating elements. The selected real constant set is used for all elements created after choosing the element attribute.

It is not needed here, but should be pointed out that Ansys uses the same concept when there are more than one material types and element types in the same analysis. The element attributes dialog box contains options for selecting element number, material number, and real constant number. Of course appropriate sets must have been defined earlier in section 2 of the input.

- Preprocessor > -Modeling- **Create**

We first create all elements that use the first real constant set. The material set number and element type number remains 1 by default.

- > **Elements** > Element Attributes > [REAL] [1] >OK

- Preprocessor > -Modeling- **Create**

- > **Elements**

- > -Auto Numbered- **Thru Nodes**

- [Node 4 and 5] > OK

- [Node 5 and 6] > OK

- [Node 6 and 7] > OK

- [Node 7 and 8] > OK

- [Node 8 and 9] > OK

- [Node 9 and 10] > OK

- [Node 10 and 11] > OK

- [Node 11 and 12] > OK

- Preprocessor > -Modeling- **Create**

We now create elements that use the second real constant set. The material set number and element type number remains 1 by default.

- > **Elements** > Element Attributes > [REAL] [2] > OK

- Preprocessor > -Modeling- **Create**

- > **Elements**

- > -Auto Numbered- **Thru Nodes**

- [Node 1 and 5] > OK

- [Node 2 and 6] > OK

- [Node 2 and 7] > OK

- [Node 3 and 11] > OK

- ANSYS Toolbar > SAVE_DB

Solution

1. Apply boundary conditions

Define zero UX and UY displacements for nodes with pin supports, and zero UY displacement for nodes with roller supports. Note each node involving beam elements has three degrees of freedom, UX, UY, and ROTZ. If you select All DOF you'll be creating a fixed support with zero rotation as well.


- Solution > -Define Loads > **Apply** > -Structural- **Displacement** > On Nodes

- Pick [Node 1] > OK > Apply U,ROT on Nodes > [UX] [UY] > OK

- (Similar steps for nodes 2 & 3. For nodes 4 & 12, [UY] only.)

Define uniform distributed load on the left span beam.

- Apply > -Structural- Pressure

- > On Beams  [Beam 2] > OK > VALI [0.2] > OK

Define concentrated loads.

- Apply > -Structural- **Force/Moment** > On Nodes > [Node 8] > OK > Direction of force/mom [FY] >

- Value [-8] > OK

- (Similar steps for nodes 9 & 10)

- ANSYS Toolbar > SAVE_DB

2. Initiate the solution

- Solution > -Solve- **Current LS**

Postprocessing

1. Plot deformed mesh and bending moment

- ANSYS main Menu > General Postproc > -Read Results- **First Set**
> Plot Results > Deformed Shape > KUND > **Def + Undeformed**
- ANSYS main Menu > General Postproc > -Read Results- **First Set**
> List Results > Element Solutions > Nodal Force > **M**

2. Look up results from the element table

Typically for each element there are a large number of quantities available in postprocessing . The exact quantities depend on the element that you are using. The information on the data available through element tables is found on-line in the Element Manual. You need to go to the section describing the element that you are using, scroll down towards the end of the element description, and there in a section titled Element Table Data you will find all data that you can access for that element. For each data set the appropriate command and option is listed.

As an example, for the 2D beam element, we might be interested in displaying the resultant shear force and the bending moment at both ends of the beams. From the Element Table Data for this element we find that the command for shear force at node I is SMISC 2, for the bending moment at node I is SMISC 8. For the purposes of display, you need to choose labels for these quantities. In the following steps, the labels **FYI** and **MI** are used for the shear force and the bending moment. The shear force and bending moment at node J are accessed by SMISC 6 and SMISC 12. For proper interpretation you must remember how you created the elements. The first node that you choose when defining an element is node I and the second one is node J. It is a good idea to follow a consistent pattern when creating elements so that it is easy to recall the order of nodes for an element.

- ANSYS main Menu > General Postproc > Element Table > Define Table
> Add
> Define Additional Element table Items
> Lab [**FYI**] > [By sequence num] [**SMISC**] > Selection [SMISC, 2]
> Apply
> Define Additional Element table Items
> Lab [**MI**] > [By sequence num] [**SMISC**] > Selection [SMISC, 6]
> Apply
> Lab [**FYJ**] > [By sequence num] [**SMISC**] > Selection [SMISC, 8]
> Apply
> Define Additional Element table Items
> Lab [**MJ**] > [By sequence num] [**SMISC**] > Selection [SMISC, 12] > OK
> Close
> Plot Elem Table > OK
>List Elem Table > Lab1-9 [**FYI**] [**MI**] [**FYJ**] [**MJ**] > OK

3. Obtain the reaction force/moment at the essential boundary points

- General Postproc > List Results > Reaction Solu > OK
- Verify that the total reaction force equals the total applied force.

Lab Problem

Use the same beam element to analyze the 2D frame shown in the figure. You may take advantage of symmetry and model a half of the frame. Use the following numerical values

$$q = 10 \text{ KN/m}, L = 2 \text{ m}, E = 210 \text{ GPa}, A = 4 \times 10^{-2} \text{ m}^2, I = 4 \times 10^{-4} \text{ m}^4.$$

Submit the following plots as part of the lab report:

- (1). The finite element mesh;
- (2). The deformed frame;
- (3). The bending moment in all the elements.

By inspection, identify the position where the bending moment reaches the maximum.

