**Structural static analysis - Analyzing 2D frame**

In this tutorial we will analyze 2D frame (see Fig.1) consisting of 2D beams with respect to resistance to two different kinds of loads:

(a) the downward shift of the point denoted by the number 5 (see Fig.1) by 5 cm,

(b) the gravity load (when gravity acceleration is $9.81\text{m.s}^{-2}$).

The cross-sectional area of all beams is $0.3\text{m} \times 0.4\text{m}$, where $0.4\text{m}$ is the height of the beams. The distance $L=5\text{m}$, the Young’s modulus is $27\text{GPa}$, the Poisson’s ratio is 0.2 and density $2800\text{kg/m}^3$.

In our implementation, we will use the element type BEAM3 and the so-called *load step files*. A *load step* is simply a configuration of loads for which a solution is obtained, i.e. you can use different load steps to apply different sets of...
loads, for example gravity load in the first load step, wind load in the second load step, both loads and a different support condition in the third load step, and so on.

1. Select the type of the discipline

ANSYS Main Menu > Preferences. We select Structural and click OK.

2. Define the type of element

ANSYS Main Menu > Preprocessor > Element Type > Add/Edit/Delete. We click Add... button, highlight Structural Mass Beam, then ‘2D elastic 3’, press OK and close the Element Types window.

Comment: Although in version 13 and 14 of ANSYS, this element has been removed from the GUI menus, it is still available by typing the command: ET, 1, BEAM3.

3. Setting the real constants

ANSYS Main Menu > Preprocessor > Real Constants > Add/Edit/Delete. We click Add... button, then OK. In the Cross-sectional area AREA we type 0.3*0.4, in Area Moment of Inertia IZZ we enter 0.3*0.4**3/12 and in Total beam height HEIGHT 0.4 (see Fig.2). We press OK and close the Real Constants window.
The same can be done by typing \( R,1,0.3*0.4,0.3*0.4^{*3/12},0.4 \) in the command line.

4. Define element material properties

ANSYS Main Menu > Preprocessor > Material Props > Material Models. We click Structural > Linear > Elastic > Isotropic and in the window that appears, we enter Young’s modulus \( E_X: 27e9 \) and Poisson’s ratio \( PR_{XY}: 0.2 \). We press OK and close the window.

ANSYS Main Menu > Preprocessor > Material Props > Material Models. We click Structural > Density and enter for density \( DENS: 2800 \), press OK and close the window.

One can see both density and linear isotropic material properties to be entered (see Fig.3).

5. Create the geometry

We will start with keypoints. Since they are equally distributed, we will define parameter \( L=5 \) to create them.

Utility Menu > Parameters > Scalar Parameters ... In Selection line we type \( L=5 \) and press Accept. We close the Scalar Parameters window.

Now we can define keypoints using parameter \( L \) according to Fig.1. We will start with the bottom keypoints and then we
will copy them. You can use either the GUI menu

ANSYS Main Menu > Preprocessor > Modelling > Create > Keypoints > In Active CS where you enter the coordinates of bottom keypoints,

or CLI by typing:

K,1,0,0
K,2,L,0
K,3,2*L,0
K,4,3*L,0
K,5,4*L,0

Then we create remaining keypoints by ANSYS Main Menu > Preprocessor > Modelling > Copy > Keypoints.

We click Pick all button and in the following window we type number of copies 5 (including the original), in Y write L and press OK, see the following figure.

Then we connect keypoints using the mouse and the GUI menu:

ANSYS Main Menu > Preprocessor > Modelling > Create > Lines > Lines > Straight Line and using the mouse we create lines. Be careful to click on every keypoint to create all 36 lines as they are depicted in Fig.4, and then click OK to close the
window.

![Fig.4: Analyzing 2D frame – lines.](image)

6. Meshing of the computational domain

ANSYS Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines. We pick all horizontal lines and enter the number of divisions NDIV to be 5, press OK. For vertical lines we follow the same way and set the number of divisions 1. Then we mesh all lines by ANSYS Main Menu > Preprocessor > Meshing > Mesh > Lines and clicking Pick All. Now all lines should be depicted in the same light blue colour.

7. Creating the Load steps files

**Load step file 1**: a downward shift of the keypoint 5 by 5cm.

To apply constrains on keypoints, we go to the Utility Menu to switch ON Numbering of keypoints: Utility Menu > PlotCtrls > Numbering ... We tick KP Keypoint numbers and click OK. Then we plot keypoints using Utility Menu > Plot > Keypoints > Keypoints.

ANSYS Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Keypoints. We pick keypoints 1, 2, 3, 4 and 5, click OK, and in the Apply U,ROT on KPs window, we highlight ALL DOF and click OK. Since no value of ALL DOF (displacements in X and Y direction and rotation) has
been given, ALL DOF equals 0.

ANSYS Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Keypoints. We pick the keypoint 5, press OK, and in the Apply U,ROT on KPs window we highlight UY (ALL DOF must be turned off). For the VALUE Displacement value we type -0.05 and press OK.

We write the Load step file 1 using ANSYS Main Menu > Preprocessor > Loads > Load Step Ops > Write LS File, type 1, and click OK (see Fig.5).

![Fig.5: The Write Load Step File window.](image)

Load step file 2: gravity load (when gravity acceleration is 9.81m.s^(-2)).

ANSYS Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Keypoints. We pick keypoint 5, click OK, and in the Apply U,ROT on KPs window we highlight ALL DOF, enter 0 in VALUE Displacement value line and click OK.

ANSYS Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Inertia > Gravity > Global. We type 9.81 in Y-direction and click OK. Note that a positive acceleration in the Y direction stimulates gravity in the negative Y direction (see a red arrow pointing in the positive Y direction in Fig.6).
Fig. 6: Load step file 2 – constrains and gravity load.

We write the Load step file 2 using ANSYS Main Menu > Preprocessor > Loads > Load Step Ops > Write LS File and type 2, click OK.

8. Running solution

ANSYS Main Menu > Solution > Solve > From LS Files. In the Solve Load Step Files window we type 1 for LSMIN Starting LS file number, 2 for LSMAX Ending LS file number and press OK, see Fig.7. ANSYS will begin solving the problem and will post a message Solution is done! when it has finished. We close the message window and go to the next step.

Fig. 7: Running solution – from Load step files.

9. Visualization and listing of results

Results can be read by ANSYS Main Menu > General Postproc > Read Results > By Pick. In the following window, see Fig.8, we highlight the first set, click Read and then Close.
Deflection:

ANSYS Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu and in Nodal Solution > DOF solution we highlight Displacement vector sum and click OK. Results after applying first kind of loads can be seen in Fig.9.

Deformation:

ANSYS Main Menu > General Postproc > Plot Results > Deformed Shape and in KUND Items to be plotted select Def + undef edge. We click OK to view both the deformed and the undeformed object. Your graphics window should look like Fig.10.
Fig.10: Load step file 1: Deformation.

Member force in the X and Y directions, and member moment about Z axis:

ANSYS Main Menu > General Postprocessor > Element Table > Define Table. We click on Add… and in Define Additional Element Table Items for Lab User label for item write MFORXA. Then in Item, Comp Results data item we highlight By sequence num, and SMISC, and enter 1 after SMISC, in the selection box. We press APPLY. In the same way we define all items according to Fig.11, where Time Stamp denotes the number of the Load step file that has been read. We click OK and close the Element Table Data window.

Fig.11: BEAM3 – element table.

To plot the items that we have defined we go to ANSYS Main Menu > General Postproc > Plot Results > Contour Plot > Line Element Results and in the Plot Line-Element Results window, and for Member force in the X direction we list MFORXA for
LabI Elem table item at node I and MFORXB for LabJ Elem table item at node J and click OK (see Fig.12). Results are depicted in Fig.13.

![Plot Line-Element Results](image1)

Fig.12: BEAM3 – The Plot Line-Element Results window.

![Load step file 1](image2)

Fig.13: Load step file 1: Member force in the X direction.

To plot Member force in the Y direction we follow the same way as before and highlight MFORYA for LabI Elem table item at node I and MFORYB for LabJ Elem table item at node J and click OK. Results can be seen in Fig.14.
Finally for Member moment about Z axis we select MMOMZA for LabI Elem table item at node I and MMOMZB for LabJ Elem table item at node J and change Fact Optional scale factor to -1 to invert the plot. The ANSYS graphics window should look similar to Fig.15.

To list items in the Element Table, we go to ANSYS Main Menu > General Postproc > List Results > Element Table Data. We highlight all MFORXA, MFORXB, MFORYA, MFORYB, MMOMZA, MMOMZB and click OK. A list, see Fig.16, with similar values should pop up. To save these results to a file, we click File within the results window (at the upper left-hand corner of this list window) and select Save as...
Now we plot and list results after applying the second load step file.

First we read results using `ANSYS Main Menu > General Postproc > Read Results > By Pick`, where we highlight the second set and click Read and Close.

Then we update the Element Table using `ANSYS Main Menu > General Postproc > Element Table > Define Table` click Update and Close.

Afterwards we follow the same way as in case of the first kind of loads. Results can be seen in the following figures.
Fig. 18: Load step file 2: Deformation.

Fig. 19: Load step file 2: Member force in the X direction.

Fig. 20: Load step file 2: Member force in the Y direction.
To obtain more precise solution, we can refine the mesh on horizontal lines and recalculate computations.