Determine the first six vibration characteristics, namely natural frequencies and mode shapes, of a structure depicted in Fig. 1, when Young’s modulus = 27e9Pa, Poisson’s ratio = 0.2, density 2500kg/m^3, and L = 6m, W = 4m, H = 4m. The frame consists of horizontal beams with the cross-sectional area 0.3m*0.6m, where 0.6m is the height of the beams, and lower vertical beams (depicted in green colour) with the cross-sectional area 0.4m*0.8m, where 0.8m is the height, and upper vertical beams (depicted in red colour) with the cross-sectional area 0.4m*0.6m, with 0.6m height. Use the element type BEAM3.

Fig. 1: Geometry of the computational domain for the modal analysis.

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: If this element cannot be found in the GUI menu, type et,1,3 in the command line.)

3. Setting the real constants

Since our structure consists of three different sets of beams, we have to define three sets of 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*0.6, in Area Moment of Inertia IZZ we enter 0.3*0.6**3/12 and in Total beam height HEIGHT 0.6. We press OK.

Afterwards in the Real Constants window we click Add…, then OK and enter Real constants for the second set:

Cross-sectional area AREA: 0.4*0.8
Area Moment of Inertia IZZ: 0.4*0.8**3/12
Total beam height HEIGHT: 0.8

In the same way we define the third set of real constants:

Cross-sectional area AREA: 0.4*0.6
Area Moment of Inertia IZZ: 0.4*0.6**3/12
Total beam height HEIGHT: 0.6.

Now your Real Constants window should look like Fig. 2. To close it, we click the Close button.
The same can be done in **CLI** by typing

R,1,0.3*0.6,0.3*0.6**3/12,0.6
R,2,0.4*0.8,0.4*0.8**3/12,0.8
R,3,0.4*0.6,0.4*0.6**3/12,0.6

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 EX: 27e9 and Poisson’s ratio PRXY: 0.2. We press **OK** and close the window.

**ANSYS Main Menu > Preprocessor > Material Props > Material Models.** We click **Structural > Density** and enter density Dens: 2500, press **OK** and close the window.

5. **Define the geometry**

We will start with four bottom keypoints:
ANSYS Main Menu > Preprocessor > Modelling > Create > Keypoints > On Working Plane where we enter the 0,0 (see Fig. 3), press Apply. In the same way we define keypoint with coordinates 6,0 press Apply, then with coordinates 10,0 press Apply, and the last bottom keypoint with coordinates 16,0 and press OK.

The same can be done by typing

K,1,0,0
K,2,6,0
K,3,10,0
K,4,16,0

in the command line.

Now we will generate keypoints on each floor:

ANSYS Main Menu > Preprocessor > Modelling > Copy > Keypoints. We click Pick All, and in the following window enter 10 in ITIME Number of copies – including original, and 4 in DY Y-offset in active CS. Click OK.

In CLI we use the KGEN command and type
In the next step, we will define lines. Again, also the GUI and mouse can be used, but we will use the *do and L command.

To create vertical lines we type

*do,i,1,4,1
*do,j,i,33+i,4
L,j,j+4
*enddo
*enddo

and to create horizontal lines

*do,i,5,37,4
*do,j,i,2+i
L,j,j+1
*enddo
*enddo

6. Assigning real constants and meshing of the computational domain

We start by selecting the horizontal lines (their numbers are in the range of 37 and 63) using the LSEL command. Then we assign attributes by the LATT command, set number of divisions to be 10 by the LESIZE command and finally mesh lines by LMES.

LSEL,s,,,37,63
LATT,1,1,1
LESIZE,all, , ,10, , , , ,1
Now we select all vertical lines by inverting our previous selection using the \texttt{LSEL,inve} command. Since we have two different kinds of vertical beams, we select (for example) the upper beams, see Fig.1, first.

\begin{verbatim}
LSEL,inve
*do,i,1,28,9
*do,j,i,i+4
LSEL,u,,,j
*enddo
*enddo
\end{verbatim}

Then we assign real constants to them by the \texttt{LATT} command, set number of divisions to be 1 by the \texttt{LESIZE} command and mesh lines by the \texttt{LMESH} command.

\begin{verbatim}
LATT,1,3,1
LESIZE,all, ,1, ,1, ,1
LMESH,all
\end{verbatim}

Finally we select the bottom beams

\begin{verbatim}
LSEL,none
*do,i,1,28,9
*do,j,i,i+4
LSEL,a,,,j
*enddo
*enddo
\end{verbatim}
and in the same way we assign attributes and mesh them. In the end we select all lines by the *LSEL,all* command.

```
LATT,1,2,1
LESIZE,all,,1,,1,,1
LMESH,all
LSEL,all
```

7. **Applying loads**

The only loads valid in a typical modal analysis are zero-value displacement constraints. If you specify any nonzero displacement constrain, ANSYS will assign a zero-value constraint to the degree of freedom instead. Also other loads can be specified, but ANSYS will ignore them. If you do not specify constraints, ANSYS will calculate rigid body modes (zero frequency).

ANSYS Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Keypoints. We pick four bottom keypoints and set ALL DOF = 0.

The same can be done by using the *DK* command.

```
DK,1,,0,ALL
DK,2,,0,ALL
DK,3,,0,ALL
DK,4,,0,ALL
```

8. **Setting the type of analysis and running solution**

ANSYS Main Menu > Solution > Analysis type > New Analysis. We tick Modal and click OK.

ANSYS Main Menu > Solution > Analysis type > Analysis Options. In [MODOPT] Mode extraction method we choose Block Lanczos. It
is a generally recommended method for large symmetric eigenvalues problems that uses the sparse matrix solver. In **No. of modes to extract** we specify the number of modes we want to extract, i.e. 6. The **[NMODE] No. of modes to expand** is usually the same as the number of extracted modes, i.e. 6 (see Fig.4). We press OK.

![Fig. 4: The Modal Analysis window.](image)

In the following window we type **100** in **FREQE End Frequency** and click OK.

At this point, we have told ANSYS to find a particular quantity of modes and to look within a particular frequency range. If ANSYS finds that quantity before it finishes the frequency range, it will stop the search. If ANSYS does not find that quantity before finishing the frequency range, then it will stop the search.

**ANSYS Main Menu > Solution > Solve > Current LS > OK.** Then we close message windows.

The list of commands for setting the type and options of modal analysis, and running the solution is

/SOLU

ANTYPE,2

MODOPT,LANB,6
MXPAND,6,,0
MODOPT,LANB,6,0,100,,OFF
solve

9. Listing and visualization of results

Natural frequencies:

We can use ANSYS Main Menu > General Postproc > Results Summary or we can type SET,LIST. A list with six frequencies, see Fig.5, will pop up. Note that each mode is stored in a separate substep.

[Fig. 5: The Results Summary with natural frequencies.]

Modes:

We turn on displaying the shape of elements using Utility menu > PlotCtrls > Style > Size and Shape and in [/SHAPE] Display of element we click ON.

Then we read results for a first substep by ANSYS Main Menu > General Postproc > First Set.

We plot deflection by 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. Your ANSYS Graphics window should look similar to the first plot of the Fig.6. We observe that the value of the maximum deflection is $DMX=0.003855$ and the value of the first frequency is $FREQ=1.545$. Then we plot deformed geometry through ANSYS Main Menu > General Postproc > Plot Results > Deformed Shape and in KUND Items to be plotted we select Def + undef edge. We click OK. Now your ANSYS Graphics window should
look similar to first plot of the Fig. 7. Again we observe the value of the first frequency and maximum deflection.

Fig. 6: Modes and corresponding natural frequencies – deflections.

To see the deformed geometry for the second substep, we read results for the second substep by ANSYS Main Menu > General Postproc > Next Set and we repeat the procedure.

Another way to read result is to use ANSYS Main Menu >
General Postproc > By Pick where we highlight the desired set and click Read button.

Fig. 7: Modes and corresponding natural frequencies – deformations.