

# Practical Training I

## *Preparation*

### 1 Beam structure

In this exercise we will work with the finite element program ANSYS. ANSYS is a widely used program for several kinds of finite element calculations like vibration analysis, thermal analysis, acoustical analysis and also—as used here—linear static analysis. Besides ANSYS there are several comparable programs available, some of them are NASTRAN, MARC, ABAQUS and the Dutch program DIANA. All these programs can perform the simple calculations that will be done in this exercise. The differences between the programs show primarily in the advanced features, in which all programs have their own strong and weak points. Most of these programs are available on a variety of computers, from PC's to supercomputers. Often, these programs will cost tens of thousands euros per year, including support by the supplier. Sometimes 'stripped' versions are available at considerably lower costs.

ANSYS is an *integrated* analysis program, which means that modelling, analysing and presentation of results takes place within one program. Specialised pre- and postprocessors like MSC/PATRAN and SDRC I-DEAS are also available on the market. These programs concentrate especially on the generation of finite element models and graphical presentation of results. They often have CAD-like capabilities and can create input files for several FEM codes. After an analysis by one of these codes, the results are read into the post-processing part of the program. A commercial license for these specialised pre- and postprocessors often costs an amount that is of the same order as the above mentioned FEM codes. Some FEM codes focus mainly on analysis tasks and have only limited pre- and post-processing capabilities. These programs are meant to be used in combination with an *external* pre- and postprocessor.

#### 1.1 The structure

This exercise considers a relatively simple frame structure like presented in Figure 1.1. The structure consists of a number of rigidly connected beams. The task is to determine the displacements of the structure and the maximum acting stress in the beams. You can prepare the exercise, based on this example.

As opposed to a truss, a beam has bending stiffness in addition to the extensional stiffness. On the right-hand side of Figure 1.1 a cross section of the beam is

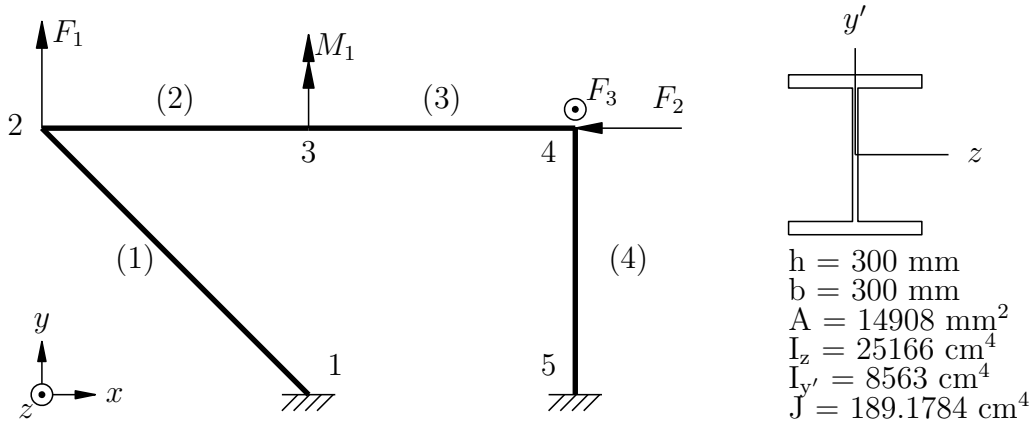


Figure 1.1: Frame structure and cross section

depicted. Here, an HE 300 B profile is used. The applicable data for this beam are referred to in the figure, where  $J$  is the torsion constant. Notice that  $J$  is not at all equivalent to the second polar moment of area  $I_x = I_z + I_{y'}$ . The Young's modulus of steel is  $210.000 \text{ N/mm}^2$  and Poisson's ratio is  $0.3$ . The height of the structure is  $5 \text{ m}$ , the distance between the supports is  $5 \text{ m}$  and the upper girder has a length of  $10 \text{ m}$ . The loads are:  $F_1 = 100 \text{ kN}$ ,  $F_2 = 20 \text{ kN}$ ,  $F_3 = 10 \text{ kN}$  and  $M_1 = 10^6 \text{ Nmm}$ .

The elements that are used for this model are beam elements. Besides displacement degrees of freedom, beam elements also possess rotational degrees of freedom. The beam element is discussed in Chapter 3 of the reader. Although the structure is two-dimensional (the practical exercise will be three-dimensional), both in-plane and out-of-plane displacements are considered and therefore a full 3-dimensional beam element is required. The 3-dimensional beam element can represent bending in two directions, torsion and extension. An element that contains all these degrees of freedom in three dimensions contains 6 degrees of freedom per node, viz. 3 displacements and 3 rotations. The simplest beam element with 2 nodes, therefore contains 12 degrees of freedom (Figure 1.2). This element

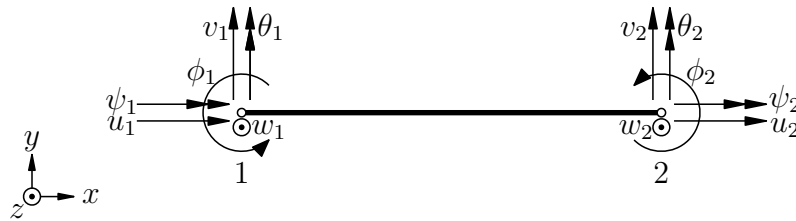


Figure 1.2: Three-dimensional beam element

has a third order polynomial displacement field perpendicular to the beam axis that is determined by the displacements  $v_1$ ,  $v_2$ ,  $w_1$ ,  $w_2$  and the rotations  $\theta_1$ ,  $\theta_2$ ,  $\phi_1$  en  $\phi_2$  in the nodes. The displacements in the direction of the beam axis and

the torsion angle follow from a linear displacement field and are determined by the displacements  $u_1$  en  $u_2$  and the rotations  $\psi_1$  and  $\psi_2$ .

The structure is divided in 4 elements. From a constructional point of view, the upper girder (between node 2 and 4) is one beam, but it is in general not possible to apply a force or moment in the middle of an element. Furthermore, a jump in the moment graph could not be represented by a third order polynomial and representing this beam in the model by two elements will increase the accuracy. In order to create the model, the following must be determined:

1. node positions
2. element types and connectivity
3. material model ( $E$  and  $\nu$ )
4. geometry of the cross section ( $A$ ,  $I_{y'}$ ,  $I_z$  en  $J$ )
5. boundary conditions (displacements and rotations suppressed in node 1 and 5)
6. external load

These aspects of modelling are general for all finite element analyses of stiffness problems and for all FEM codes. Only the geometry of the cross section is particular for beam elements.

One of the first decisions to take in the modelling phase is the choice of a suitable system of units. Make a choice and stick to that. This choice does not only affect the geometry (m, cm, mm, inch etc.) but also the material parameters (N/m<sup>2</sup> or N/mm<sup>2</sup>). If gravity is included it will become even more complex. In such cases it is useful to try with a small model whether the program reacts as expected.

In the next section we will consider in some detail how to model a frame structure within ANSYS.

## 1.2 Modelling with ANSYS

Within ANSYS the pre-processor part is used to generate a model. When ANSYS is started a window pops up as indicated in Figure 1.3. Underneath a small description is given; during the practical exercise a more elaborate instruction will be available.

It is important to distinguish the parts of the window. Firstly, at the top, we find the *Utility Menu*. With this menu all kinds of settings can be changed and commands be given. We will use it especially to adapt the settings for plotting of the model and the results with the 'PlotCtrls' selection. For example, the selection of plotting of nodes and/or elements with or without node and element numbers is determined with this menu. A very important item in the utility menu is the *Help* item. With this choice the complete ANSYS manual can be viewed on-line.

Below the *Utility Menu* we find an input box. In this box, commands can be typed verbatim. Experienced users can execute certain actions in this way, usually in

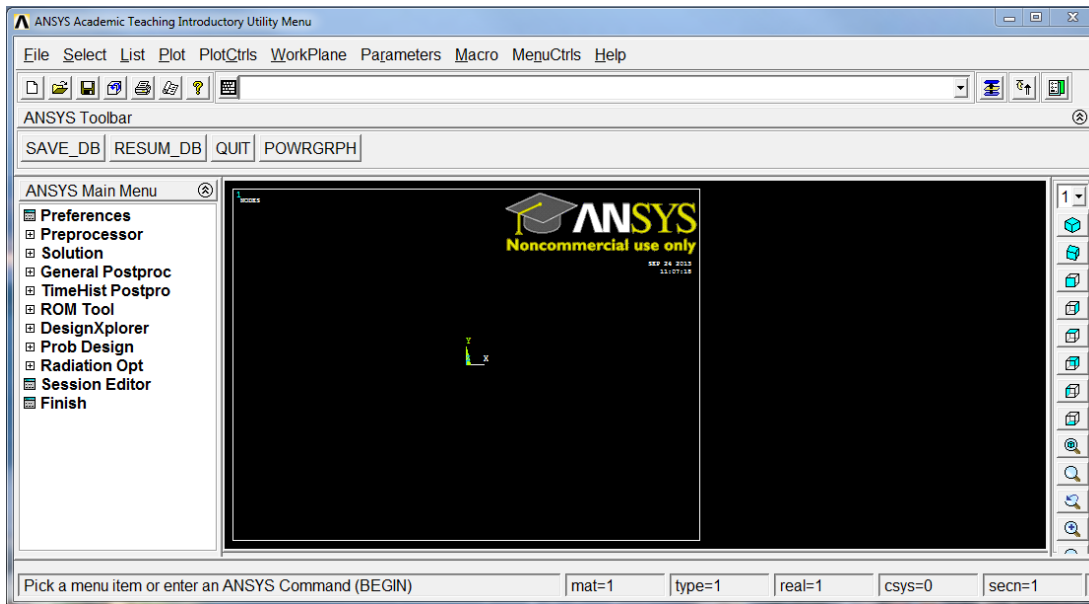


Figure 1.3: ANSYS start windows

less time than by menu choices and mouse clicks. The use of the commands is, however, not obvious and we will not use it here. Below the input box, we see the *Toolbar* (if it is not visible, click on *MenuCtrls* » *Update Toolbar* in the *Utility Menu*). This box is used to execute high level commands with a single mouse click. Saving and restoring the complete model is the most important usage.

On the left-hand side we find the *Ansys Main Menu*. Although the commands that can be given in this menu can also be reached through the *Utility Menu* and the *Input* box, the Main Menu is mostly used because the sequential steps are presented in a logical order. Especially the *Preprocessor*, *Solution* and *General Postproc* submenus are of importance.

A large part of the screen is reserved for the *Graphics* window. In this window, the model and the analysis results are presented. It starts empty. Underneath all these screens is the *Output* window. On this window, the commands that ANSYS receives are echoed and messages are given about what the program is doing. From this window it is clear that the graphical user interface is a shell around a kernel that works with a simple command structure. In general, this window is not needed, but if something goes wrong, you may find relevant messages in this window.

The most important sub-menus from the *Main Menu* are presented in Figure 1.4. Most selections will pop up other menus or dialogs. The depicted sub-menus represent the general process flow of a FEM analysis: creating a model (pre-processing), performing the calculation (solution) and interpretation of results (post-processing).

To input a model you click on *Preprocessor* in the *Main Menu*. This will give the choices as presented in Figure 1.4a. From top to bottom a number of sub-menus are selected. Some sub-menus are not necessary for this exercise. The first three

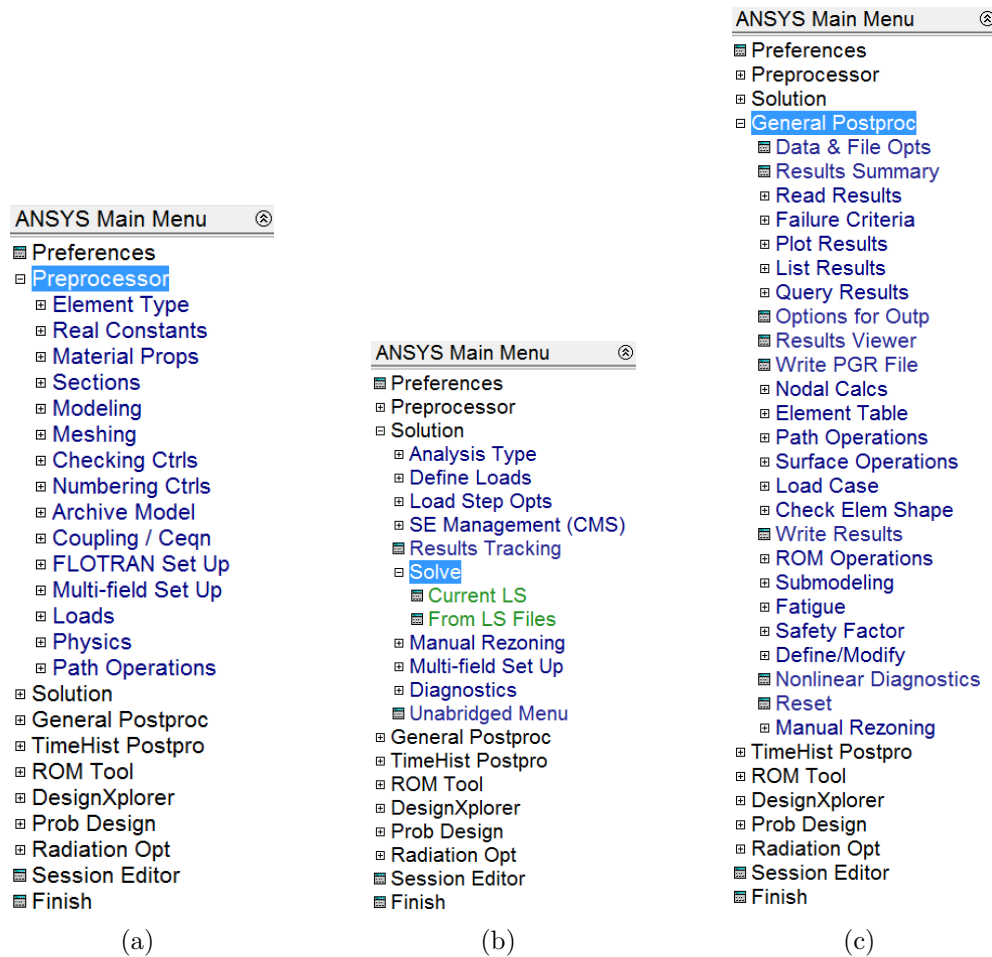


Figure 1.4: Sub-menus of the ‘Main Menu’

items are only preparatory, they do not really create a part of the model, but initialise data to be referred to later. This is an important concept to keep in mind. It is useful to determine in advance which material types appear in the structure, which cross sections etc. These items can then already be prepared for later reference. In our example e.g. from the more than 100 available element types an element type must be selected. On top of that, a number of options per element type may have to be selected.

The *Real Constants* contain data about e.g. thickness and second moments of area (typically the data on the right-hand side in Figure 1.1). These quantities are also stored for later reference. Finally we must create the material references such that, later, we can refer to e.g. material number 1. In this example, we must attribute to material number 1 the parameters for the steel beams.

After these preparations we must use the *Create* menu (within the *Modeling* menu) to create the coordinates of the nodes and the elements. Automatically material number 1 and real constants number 1 will be assigned to every created element, unless explicitly selected otherwise. In the current example, the items under *Meshing* are not used, because all nodes and elements are added manually.

Finally, we only need the *Loads* item. With this selection, external loads (forces and moments) are supplied, as well as supports or prescribed displacements and rotations. During data input, the model is presented in the graphics window, so you can check whether the model develops as intended. The supports and loads are also depicted.

After creation of the model you can close the pre-processor sub-menu and the analysis can be performed by selecting the *Solution* menu in the *Main Menu*. In the *Solution* sub-menu (Figure 1.4b) you must select *Current LS* (Load Step) under *Solve*. Follow the directions and be on alert for messages that indicate that something is wrong with the model. If these are absent, you can click away the */STATUS Command* window of the solution phase and view the results with the post-processor (see Figure 1.4c). Two sub-menus are especially important. *List results* for numerically viewing results and *Plot results* for a graphical presentation of results. Note that some of the sub-menus are not visible if the model has no results. In the next section the interpretation of results is considered.

### 1.3 Elaboration of results

When the solution phase is performed we can start with the elaboration of the results. This elaboration is split in two parts. First, the validity of the results must be assessed. Only if the validity of the analysis is confirmed it makes sense to interpret the results for the purpose of the analysis. Take care that conclusions are drawn only about features that can really be represented by the applied model. The validity check of the results can be done in different ways. A list with checks is given below.

1. Are the external loads and reaction forces in equilibrium?
2. Does the magnitude of the displacements agree with ‘engineering judgement’ and are the displacements small with respect to the thickness of the beams (otherwise a nonlinear analysis would have to be performed)?
3. Does the deformed shape ‘look good’ (direction in agreement with the main loads)?
4. Does the displacement agree with the experienced load? Check e.g. for element 4 whether the displacement of node 3 is in agreement with the basic bending relations. In this way you check amongst others whether material data and second moments of area are correctly represented.
5. In cases where a manual calculation is not obvious: make a (strongly) simplified model and analyze *that* model manually in order to get an impression of the magnitude of displacements and/or stresses. Check whether the FEM results are of the same order of magnitude.

For the first check we can use the command to print reaction forces. We then get the following information:

PRINT REACTION SOLUTIONS PER NODE

\*\*\*\*\* POST1 TOTAL REACTION SOLUTION LISTING \*\*\*\*\*

LOAD STEP= 1 SUBSTEP= 1  
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING X,Y,Z SOLUTIONS ARE IN GLOBAL COORDINATES

NODE	FX	FY	FZ	MX	MY	MZ
1	72358.	-.11518E+06	-110.24	-.61832E+06	-.51679E+06	.14069E+09
5	-52358.	15177.	-9889.8	-.49382E+08	68002.	.18343E+09

TOTAL VALUES

VALUE	20000.	-100000.	-10000.	-.50000E+08	-.44879E+06	.32412E+09
-------	--------	----------	---------	-------------	-------------	------------

From the summed totals it is easily seen that the reaction forces are in equilibrium with the applied loads. For the equilibrium of moments this is not directly clear. Check the equilibrium of moments yourself, considering also the reaction forces. It will turn out to be correct.

For the magnitude of the displacements we can print all displacements and rotations. We then obtain:

PRINT DOF NODAL SOLUTION PER NODE

\*\*\*\*\* POST1 NODAL DEGREE OF FREEDOM LISTING \*\*\*\*\*

LOAD STEP= 1 SUBSTEP= 1  
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN GLOBAL COORDINATES

NODE	UX	UY	UZ	ROTX	ROTY	ROTZ
1	.00000	.00000	.00000	.00000	.00000	.00000
2	22.976	23.400	.75464	.24642E-02	-.22346E-02	-.45010E-02
3	22.861	12.274	11.824	.46601E-02	-.21675E-02	-.11458E-02
4	22.745	-.24239E-01	22.869	.68561E-02	-.22252E-02	-.49701E-02
5	.00000	.00000	.00000	.00000	.00000	.00000

MAXIMUM ABSOLUTE VALUES

NODE	2	2	4	4	2	4
VALUE	22.976	23.400	22.869	.68561E-02	-.22346E-02	-.49701E-02

We see a maximum displacement of approximately 23 mm in all directions and rotations of the order of  $10^{-2}$  radians, or about a half degree. The force  $F_1$  is comparable with a weight of approximately 10 metric tons and the beams are typical for the columns in the lattice of a 10 story building. Twenty-three millimetres then seems to be a reasonable displacement. On a cross section height of 300 mm the displacement is not very small, but still within limits. As a rule of thumb one tenth of the height is considered acceptable. Above that value geometrically nonlinear effects become relevant, although the relevance strongly depends on the shape of the structure and the boundary conditions.

Thereafter, we consider the deformed geometry from different view angles. ANSYS scales the displacements, because otherwise the deformations would hardly be visible. In Figure 1.5 two views are presented. In the figures, the deformed

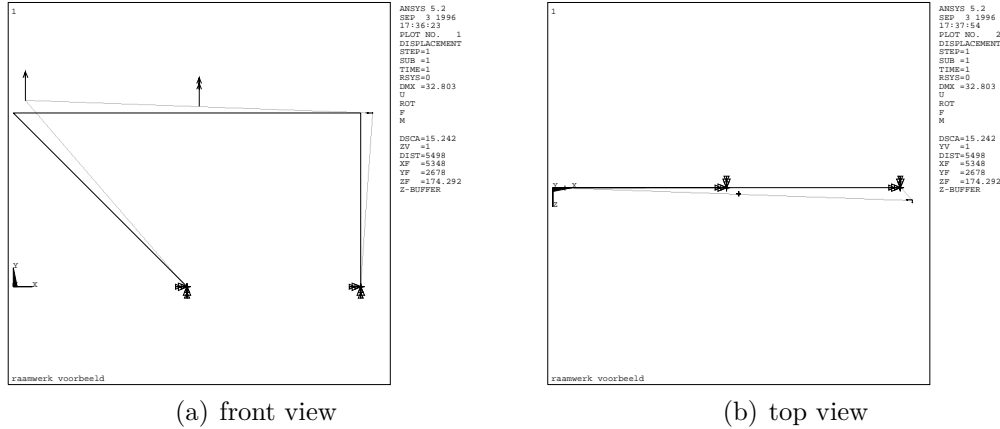


Figure 1.5: Displacements

and the undeformed geometry are presented. On screen they will have different colours. The deformations in the front view look reasonable. In the top view, however, it seems that there is no deformation due to moment  $M_1$ . A closer look reveals that  $10^6$  Nmm is a large number true enough, but that the influence of  $F_3$  is much larger, since this delivers with respect to node 3 a moment of  $5 \cdot 10^7$  Nmm.

Finally, we will check for element 4 whether the deformations are correct. This is for beam elements especially relevant to check if the  $y'$ - and  $z$ -axis of the cross section are correctly represented and not accidentally swapped (see Figure 1.1). Therefore we output the element forces:

PRINT FORC ELEMENT SOLUTION PER ELEMENT

\*\*\*\*\* POST1 ELEMENT NODE TOTAL FORCE LISTING \*\*\*\*\*

LOAD STEP= 1 SUBSTEP= 1  
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING X,Y,Z FORCES ARE IN GLOBAL COORDINATES

ELEM=	1	FX	FY	FZ	MX	MY	MZ
1	-72358.	.11518E+06	110.24	.61832E+06	.51679E+06	-.14069E+09	
2	72358.	-.11518E+06	-110.24	-67108.	34426.	-.73406E+08	

ELEM=	2	FX	FY	FZ	MX	MY	MZ
2	-72358.	15177.	110.24	67108.	-34426.	.73406E+08	
3	72358.	-15177.	-110.24	-67108.	-.51679E+06	.24788E+07	

ELEM=	3	FX	FY	FZ	MX	MY	MZ
3	-72358.	15177.	110.24	67108.	-.48321E+06	-.24788E+07	



	4	72358.	-15177.	-110.24	-67108.	-68002.	.78364E+08
ELEM=	4	FX	FY	FZ	MX	MY	MZ
	4	-52358.	15177.	-9889.8	67108.	68002.	-.78364E+08
	5	52358.	-15177.	9889.8	.49382E+08	-68002.	-.18343E+09

For the load on element 4 we may of course not only use  $F_2$  and  $F_3$ , the forces and moments that are acting from element 3 on element 4 must also be considered. We therefore take the (calculated) forces and moments that ANSYS gives for element 4 and node 4. In the first place, the extension must fulfil  $\varepsilon = \sigma/E = F_y/AE$ . The displacement in  $y$ -direction of node 4 is then  $u_y = \varepsilon \cdot l = \frac{F_y l}{AE} = \frac{15177 \cdot 5000}{14908 \cdot 210000} = 0.024239$  mm. This agrees with the calculated value UY. A similar relation holds for torsion and the moment  $M_y$ :  $\psi = \frac{M_y l}{GJ} = \frac{68002 \cdot 5000}{80769 \cdot 1891784} = 0.0022252$  where the relation  $G = \frac{E}{2(1+\nu)}$  was used. The value for  $\psi$  matches exactly the value of ROTY in ANSYS for node 4. The displacement in  $x$ -direction is calculated with basic bending equations. With the force  $F_x$  and moment  $M_z$  we find:

$$u_x = \frac{F_x l^3}{3EI_z} - \frac{M_z l^2}{2EI_z} \quad (1.1)$$

Notice the sign; a positive moment  $M_z$  gives a negative displacement  $u_x$ . After substitution this gives  $u_x = -22.7448$  mm. Within numerical accuracy, this agrees with the result from ANSYS. The displacement in  $z$ -direction follows from:

$$u_z = \frac{F_z l^3}{3EI_{y'}} + \frac{M_x l^2}{2EI_{y'}} \quad (1.2)$$

After substitution this yields:  $u_z = -22.869$  mm. This also agrees with the ANSYS results. Apparently  $I_{y'}$  and  $I_z$  are represented correctly.

Note that ANSYS has found the ‘exact’ solution. This should be the case, since the ‘real’ displacement field (third order polynomial) can be described exactly with the ‘approximating’ displacement field. Remember, however, that also the basic bending equations are just an approximation for the physical behavior, in which it was assumed that a plane perpendicular to the neutral line remains plane and perpendicular. The exact deformation field in the vicinity of supports and connections cannot be described with simple bending equations and hence neither with this finite element model.

Now that we feel reasonably sure that the analysis is correct, it is useful to elaborate the results further. We can e.g. determine the maximum and minimum stress in different cross sections of the beams. A selection of the results that you get is:

PRINT ELEM ELEMENT SOLUTION PER ELEMENT

\*\*\*\*\* POST1 ELEMENT SOLUTION LISTING \*\*\*\*\*

LOAD STEP        1    SUBSTEP=        1  
TIME=        1.0000        LOAD CASE=    0

```
EL=      1  NODES=      1      2  MAT=   1
LOCATION   SMAX          SMIN
   1 (I)   94.156      -76.366
   2 (J)   52.689      -34.898
```

```
EL=      2  NODES=      2      3  MAT=   1
LOCATION   SMAX          SMIN
   1 (I)   38.960      -48.667
   2 (J)  -2.4709      -7.2364
```

```
EL=      3  NODES=      3      4  MAT=   1
LOCATION   SMAX          SMIN
   1 (I)  -2.5297      -7.1776
   2 (J)   41.973     -51.681
```

```
EL=      4  NODES=      4      5  MAT=   1
LOCATION   SMAX          SMIN
   1 (I)   45.807     -47.844
   2 (J)  194.82     -196.85
```

From this it appears that the largest stress is equal to  $196.85 \text{ N/mm}^2$  (compressive stress). It is not enough to look at the maximum stress given by ANSYS because the minimum stress can be larger in magnitude than the maximum stress. Most steels will be able to resist this stress elastically. It depends on the required safety factors whether this stress is admissible.

After the first analysis, some variations can be considered.

1. Replace the fixed supports in node 1 and 5 by simple supports that can rotate along the  $z$ -axis.
2. Increase the moment  $M_1$  to  $100 \text{ MNmm}$ . The top view must now get an "S"-shape. This will however not be seen in an ANSYS plot, because the 2 end nodes will be connected (graphically) by a straight line. Replace, therefore, the upper 2 elements by 6 elements. Note that this does not affect the accuracy of the results (two elements are already 'exact').
3. Replace the material of element 4 by concrete:  $E = 30000 \text{ N/mm}^2$  and  $\nu = 0.2$  and a square cross section of  $500 \times 500 \text{ mm}^2$ . The second moment of area can be calculated to be  $I_z = I_{y'} = \frac{1}{12}bh^3$  and  $J = 0.84I_p = 0.84(I_z + I_{y'})$ .

When the results are interpreted, it must still be kept in mind what the model represents and what can be expected from the model. In this case, a linear analysis is considered, therefore the displacements and stresses will be proportional with the loads. Buckling of the structure cannot be analysed in this way. If the load is taken 10 times higher, the displacements will be 10 times higher. In other words: *in a linear analysis you will never predict buckling*. In our case, the structure will probably not be sensitive to buckling in the  $xy$ -plane (at this load). Whether the structure can also resist buckling in the  $z$ -direction remains to be seen and cannot be determined from this analysis.