

Practical Training II

Instructions

1 Introduction

In this document, a step by step instruction is given for the use of ANSYS in practical exercise 2 of the course ‘Introduction to the Finite Element Method’. For a more comprehensive description of several commands we refer to the on-line ANSYS user’s manual (click on Help in the *Utility Menu*).

A selection from one of the menus is depicted with a frame around the command. Before starting the exercise you should make a separate directory to store your ANSYS files.

2 Installation and start-up

ANSYS is a commercial finite element package which is used by both researchers here at the University of Twente, as well as in engineering companies. As a student, you can get a limited version at the ANSYS website. The number of structural element/nodes will be limited to 32.000 with this license, which is more than enough for the tutorials. If you download the paid licensed version of ANSYS via the UT Software Manager, the University might run out of licenses and you will not be able to start ANSYS and complete the tutorials.

The basis of ANSYS is the APDL (Ansys Parametric Design Language). In this environment, the model must be built manually. More recently, ANSYS workbench has been developed, which is a more user-friendly GUI around the APDL, aimed to be an intuitive workflow program in which CAD-files can be imported. This tutorial we will be using ANSYS Mechanical APDL.

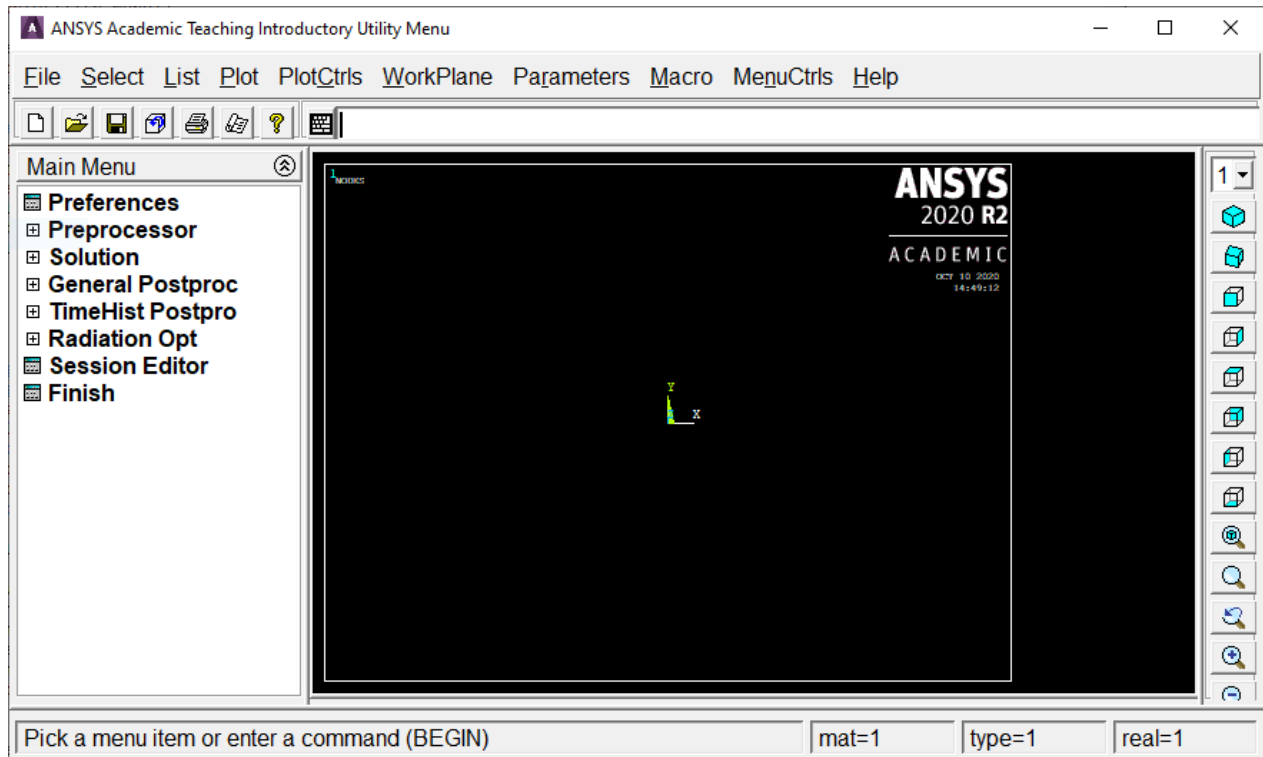


Figure 1.1: ANSYS start window

Start an interactive ANSYS-job by opening `Mechanical APDL [version]`. You can also start a session by using `Mechanical APDL Product Launcher [version]` and start the APDL directly from a specified working directory. A window as presented in Figure 1.1 will appear. When not started from the product launcher, make a working directory via `File`, `Change Directory...` in the *Utility Menu*.

The general sequence of events in a FEM analysis starts with generating the FEM model (preprocessing) then the actual calculation (solution) and finally the elaboration of the results (post-processing). These aspects take place in different parts of the ANSYS program. We will find them underneath each other in the *ANSYS Main Menu*. They are discussed in the following sections. In between the section on the preprocessing and solution phases, a section about the numerical and graphical presentation of model data is included.

3 Preprocessor

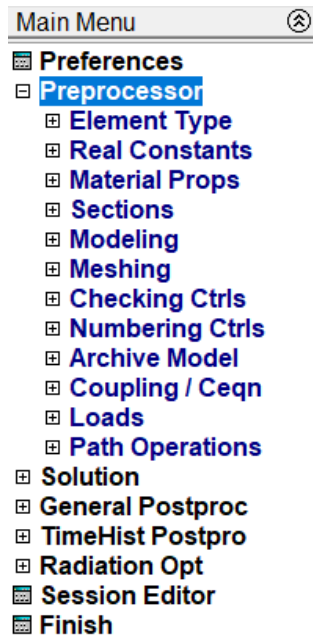


Figure 1.2: Preprocessor

We will now first create a model with the Preprocessor. For that purpose, select from the *Main Menu* the item **Preprocessor**. This results in a window as presented in Figure 1.2.

In the Preprocessor data is prepared and stored for later reference ([ELEMENT TYPE], [REAL CONSTANTS], [MATERIAL PROPS], [SECTIONS]), a mesh is generated and a load applied. In this exercise, the mesh is created by manually adding the position of the nodes and connecting elements to these nodes.

The loads consist of prescribed displacements and rotations (that may be equal to zero—the supports) and of loads like forces and moments.

N.B. Data is often entered using 'pop-up' windows. Sometime you might 'lose' such a window. You can retrieve such a window by pressing on the *Raise Hidden* button at the right side of the *Command Prompt* box.

3.1 Initialisation

First, a number of items are prepared and stored for later reference. These are the items, discussed above ([ELEMENT TYPE] to [SECTIONS]) in Figure 1.2. They are—from top to bottom—to be used element types, constants like cross section areas and

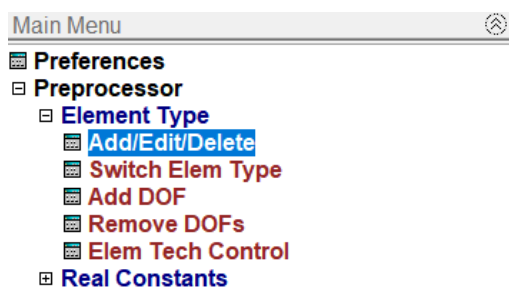
finally material data.

Note that we are not yet creating a model, but only preparing data that will be used later when we really create a model.

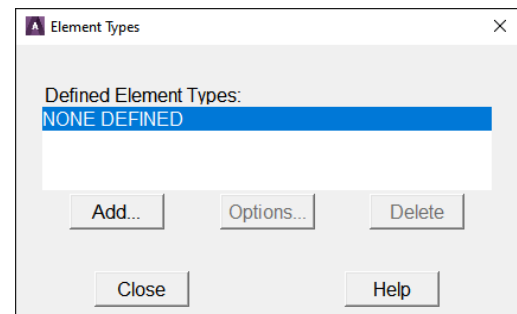
After selection of the several items, more (sub)menus or forms appear.

Element type

In Figure 1.3a the window is presented that is obtained by opening the sub-menu **Preprocessor**, **Element Type**. By clicking on **Add/Edit/Delete** new elements can be added to the selection. A window appears as given in Figure 1.3b.



(a)



(b)

Figure 1.3: Element type

By clicking on **Add** the *Element library* window (Figure 1.4) appears. In this window, select the 3D beam element by first selecting **beam** and then **2 node 188**. After clicking **OK** we shall see that element type BEAM188 is selected under reference number 1.

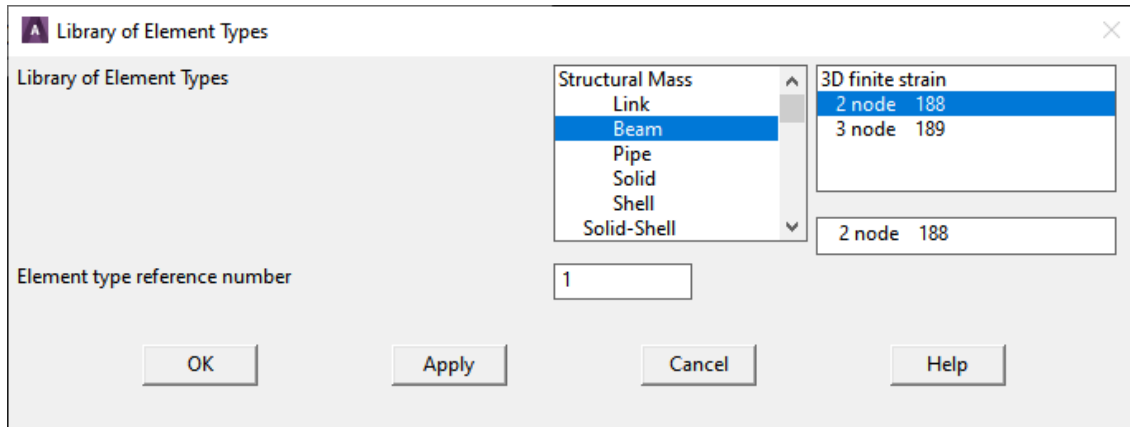


Figure 1.4: Element library

Some options of the element should be changed. Click on **Options...** and change the *Cross section scaling is K2* to **Rigid (classic)** and the *Element behavior K3* to **Cubic Form.** (Figure 1.5).

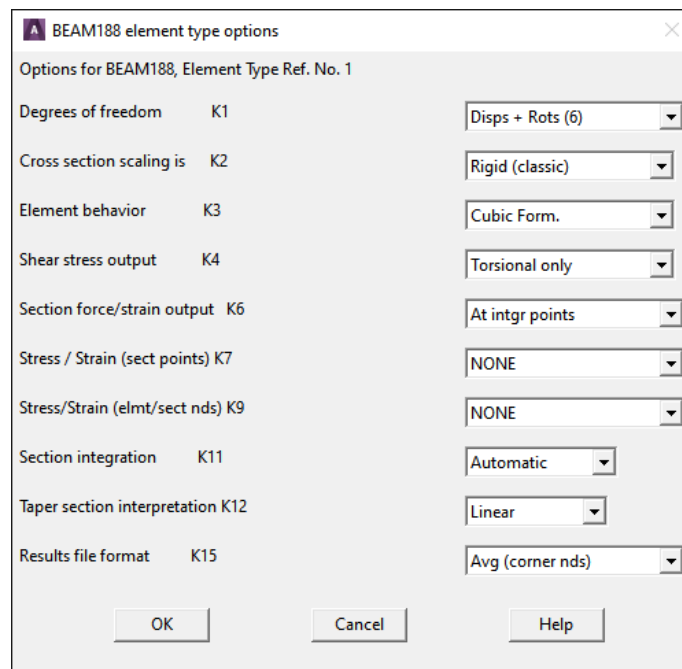

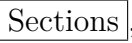



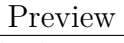


Figure 1.5: Element type options

Since this is the only element type that will be used in this exercise, this is enough for the element selection. The *Element Types* window (Figure 1.3b) can be closed with the **Close**



button. Minimise the sub-menu of *Element Types* in the *ANSYS Main Menu*, by clicking on the  besides **-Element Types-**. This keeps the *ANSYS Main Menu* clearly structured.

Sections

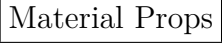
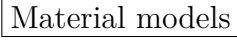
Now we have to define the cross-sections of the beams. Click on , , . Select the square cross-section at *Sub-Type*. Set B and H to the values given in the exercise. Set both Nb and Nh to 1. These last two values set determine in what way the stress results will be available for post-processing. Now click on . You can see the cross-section and values of the moments of inertia by clicking on . You will need these values while comparing the displacements of one element with the analytical solution.

Note that it is very inefficient to build a construction with rectangular solid beams. However, we choose this option in this exercise for simplification of the post-processing.

For the second cross-section you should change the *ID* to 2 and enter the corresponding values. Click on  and .

Go to  and change *TXZ* and  to $1E12$. Do this for both sections.

Material Properties

The last preparatory action is the definition of the applied material model. By clicking on the sub-menu  and subsequently on  a window as in Figure 1.6 appears.

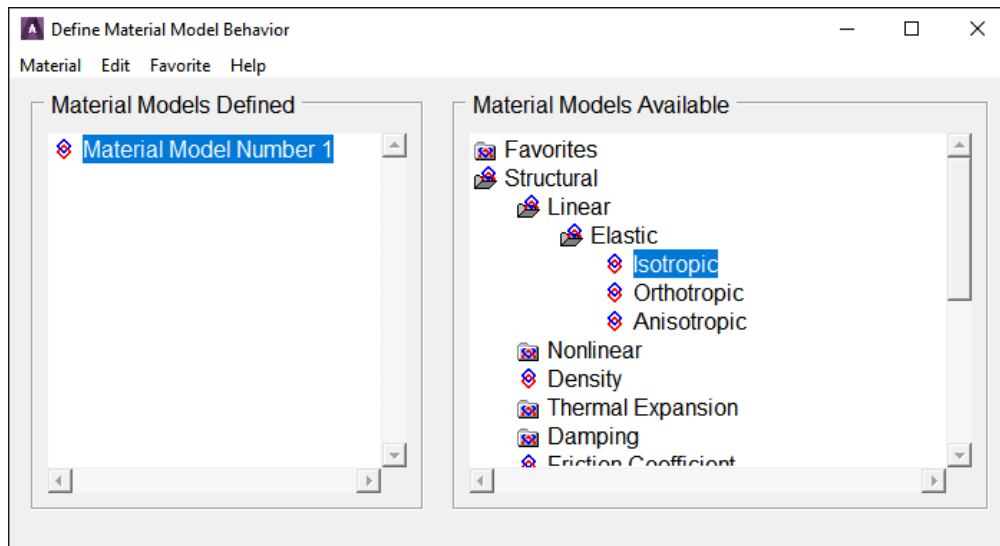






Figure 1.6: Define material model behaviour

In this window, several types of material models can be selected. For this exercise we can apply constant isotropic elastic material behaviour (independent of the loading direction). Therefore, we select , , , . We now obtain a window as given in Figure 1.7.

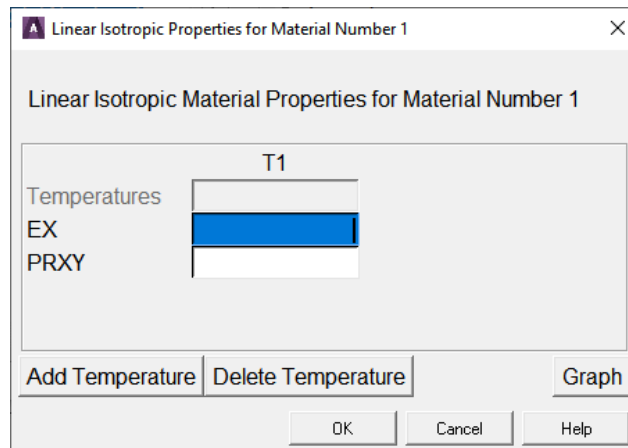


Figure 1.7: Linear isotropic properties for material number 1

The material with reference number 1 can be defined here. At least the Young's modulus (EX) and Poisson's ratio (PRXY) must be filled in. Note that the applied units must match the units that are used to describe the complete structure (m, mm, etc.). Click on **OK** and close the window as presented in Figure 1.7 to return to the *ANSYS Main Menu*. Now, sufficient data is prepared and stored in order to start the creation of the model.

3.2 Nodes and elements

We will now determine the placements of nodes and elements. For the simple case of this exercise, the coordinates of the nodes will be defined manually.

For that purpose, in the Preprocessor, click on **Modeling** (Figure 1.2) and subsequently on **Create**. We will now first create the nodes by clicking on **Nodes** and then on **In Active CS**. A window as in Figure 1.8 appears.

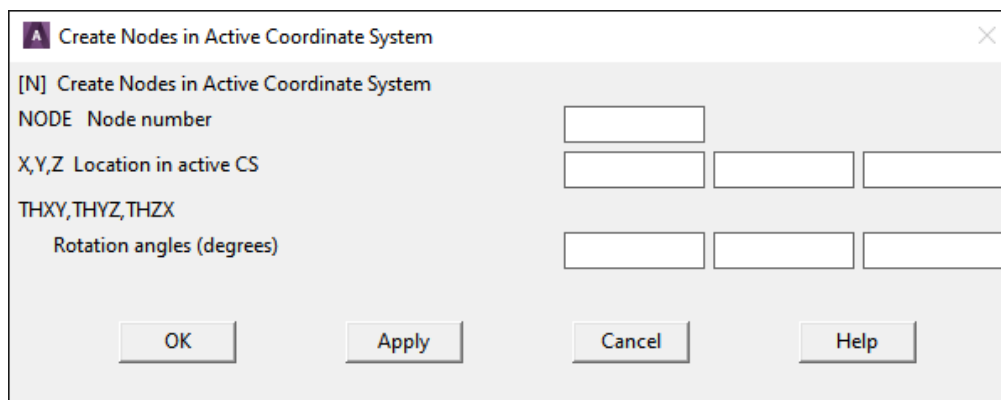


Figure 1.8: Creation of nodes

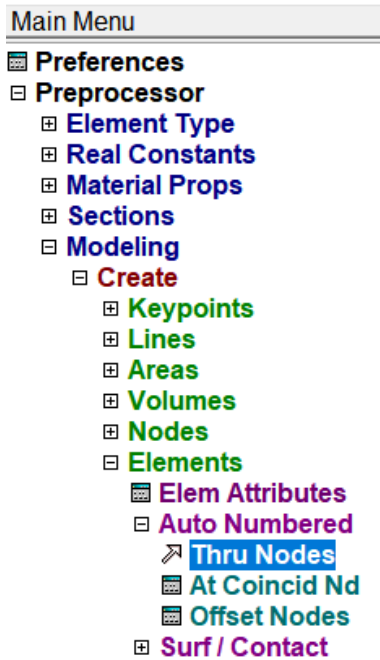


Figure 1.9: Create Elements

We can now fill-in the node numbers and coordinates (in mm). You can leave the *rotation angles* empty. By clicking on **Apply** instead of **OK** the window stays open and the next node can be created. Do this until all required nodes are defined and close with **OK**.

Subsequently, the elements will be connected to the nodes. Minimise the *Nodes* menu and select **Elements**. We will connect the elements directly to the nodes and leave the element numbering to ANSYS. Choose therefore **Auto Numbered**, **Thru Nodes** in the *Elements* menu in Figure 1.9. In the graphics window, we can (with the special arrow) click on two nodes and then on **Apply** to create one element. Repeat this until all elements are created and close the input with **OK**.

We must now take care that the correct *Sections* (the cross section data) are attributed to the correct beams. This can be done by subsequently clicking on **Modeling**, **Move/Modify**, **Elements**, **Modify Attrib**. Then click on the elements that should refer to *Section* set 2. After

Apply a window appears as presented in Figure 1.10.

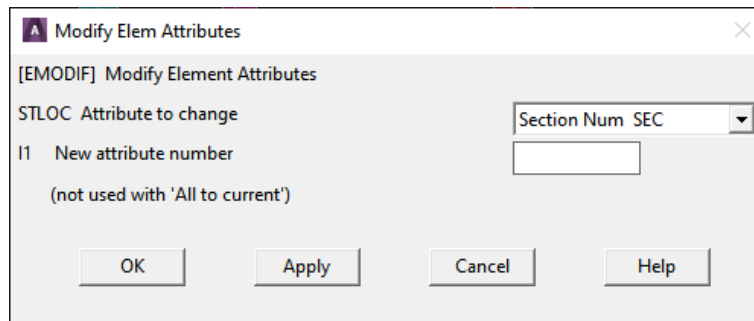


Figure 1.10: Modify Elem Attributes

Select in the upper box **Section Num SEC** and in the lower box 2. Check with the *Utility Menu* **List** **Elements** **Nodes + Attributes** whether the elements refer to the correct material and real constants sets.

When the nodes in the graphics window are (no longer) visible, switch on the node numbers by clicking in the *Utility Menu* on **PlotCtrls** and then on **Numbering ...**. Check the box behind *NODE*. Another plotting setting has to be changed to be able to plot the stresses as a contour plot during post-processing. Go to the *Utility Menu* and click on **PlotCtrls**, **Style**, **Size and Shape...**. Check the box behind *[/ESHAPE] Display of element*. Thereafter, use the *Utility Menu* **Plot** **Nodes** to plot the nodes and **Elements** to plot the elements. With this, the geometry of the model is defined.

3.3 Supports and loads



Figure 1.11: Loads

The supports and loads, both fall within the range of the item **Loads** in the Preprocessor menu (Figure 1.2). Open the following sub-menus subsequently: **Loads**, **Define Loads** and **Apply**. Below this menu selections for prescribed displacements or external forces and moments can be made.

The supports must be prescribed by setting the displacements to zero. For that purpose, open the sub-menu **Displacement**. Select that you want to suppress displacements of nodes by checking **On Nodes**. Subsequently click on the nodes of which you want to suppress the same set of degrees of freedom (displacements and/or rotations). When you finally click on **Apply** or **OK** you will obtain a window as in Figure 1.12. In the large window, select the degrees of freedom that must be suppressed and set the VALUE to '0'. In the graphics window the supports are now presented. If they disappear later, they can be made visible again by selecting in the *Utility Menu* **PlotCtrls** and subsequently **Symbols**. Then check the 'All BC+Reaction' (B.C.) box to make the boundary conditions visible.

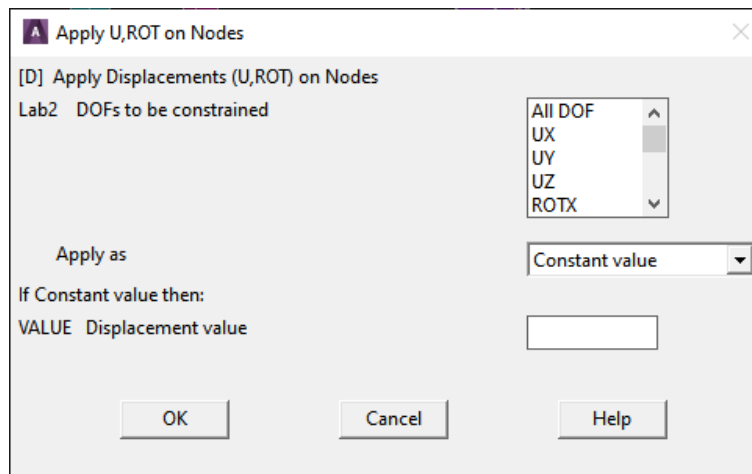


Figure 1.12: Prescribed displacements

Leave the *Displacement* menu.

We finish the creation of the analysis model by applying the external loads. Therefore, select the menu item **Force/Moment** (low in Figure 1.11).

Since we want to prescribe forces and moments on nodes, we select **On Nodes**. Now click in

the graphics window on the nodes on which the same force and moments act (probably only one here). If you click on **OK** or **Apply** you will obtain a window as presented in Figure 1.13.

Select the required load (force or moment in x -, y -, or z -direction) and fill-in the magnitude. Note again that this must be consistent with the units that are used for the geometry and the material (e.g. Nm or Nmm).

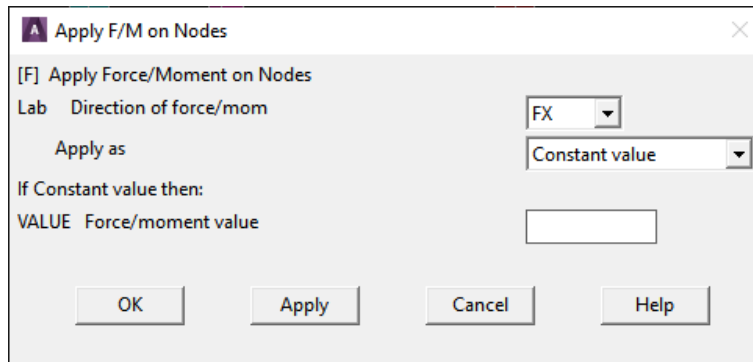


Figure 1.13: Force and Moment creation

The complete analysis model is now created and the **-Preprocessor-** sub-menu can be minimised (click on **☐** next to **-Preprocessor-** in the *ANSYS Main Menu*).

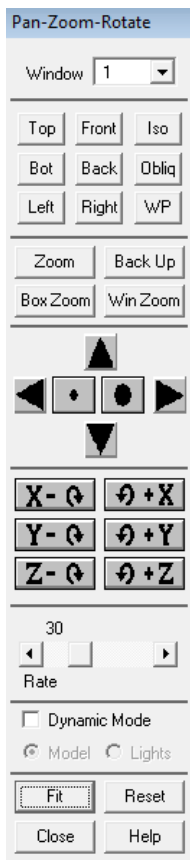


Figure 1.14: Pan, Zoom, Rotate

4 Model checks

After generation of the model (or parts thereof) a model can be checked numerically and graphically. This is done with the options **List**, **Plot** and **PlotCtrls** in the *Utility Menu*. With the 'List' options, several quantities can be presented numerically in a separate window. With the 'Plot' options the same data can be presented graphically in the graphics window. With the 'PlotCtrls' menu the graphical presentation can be changed.

A not yet described option is to change the size of the picture, zoom in or plotting from a different view point. This can be done with selections in the **Pan, Zoom, Rotate** form from the 'PlotCtrls' menu. In that case, a window as in Figure 1.14 appears. Just try the buttons in this window and look what happens with the presentation of the model. This also applies later to the presentation of the results. You can select **Reset** in order to restore the initial situation.

With the **Numbering** item in 'PlotCtrls' you can select whether for nodes, elements etc. numbers must be included in the graphics window. With the **Symbols** item in 'PlotCtrls' you can select whether applied supports or loads must be plotted. Sometimes, after a change of options an explicit **Plot**, **replot** command must be given in the *Utility Menu*.

5 Solution

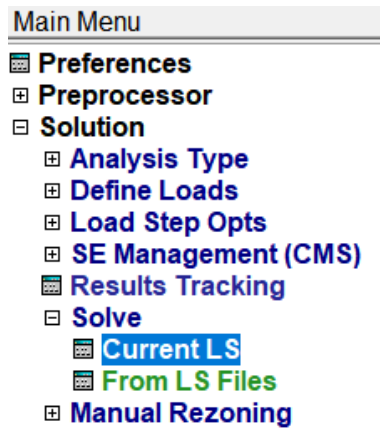


Figure 1.15: Solution

When the model is completely determined and checked, the analysis can be started. For this purpose, open in the *Main Menu* the sub-menu **Solution** (see Figure 1.15). In order to start the analysis you must click on **Solve**, **Current LS**. If, in the preceding preprocessing, elements were created and deleted, sometimes an error message appears about the appearance of an element that does not show up with the **List** command. This is an apparent error in the program that can be solved by using, in sub-menu **Preprocessor**, **Numbering Ctrl**s (see Figure 1.2) the option **Compress Numbers**. Then select 'elements' and if required then once with 'nodes'. Internally, the unused element and node numbers are removed.

6 General Postproc



Figure 1.16: PostProcessing

When the analysis is executed, we can view the results. Firstly, the validity of the results must be assessed and thereafter displacements and stresses can be compared with the design specifications.

Just try the variety of results that can be presented. Make a copy of the database first with the **SAVE_DB** button in the *Toolbar*. If necessary, the database can be restored with the **RESUM_DB** button.

When we select the postprocessor from the *Main Menu* by selecting the sub-menu **General Postproc**, we will get a window as in Figure 1.16. Since we only performed a single statical analysis, we do not have to select results or read them separately. However, in some cases an error arises when the stress results are requested. In that case it is required to click on **Last Set** to read in the results. We will only use the sub-menu **Plot Results** and sub-menu **List Results**.

In the sub-menu **Plot Results**, the deformed shape of the construction can be plotted by clicking on **Deformed shape**. Furthermore, contour plots of the nodal and element solutions can be made. For example, click on **Contour Plot**, **Element Solu**,

Element Solution, **Stress** and **von Mises stress** to see the stress distribution in the construction.

With numerical output in the **List Results** sub-menu, nodal and element results are relevant. With **Nodal Solution** the value of the several degrees of freedom can be presented (Dof Solution). Have a look at the displacements as well as the rotations.

In **Element Solution**, the item **All available force items** can be selected. With this item,

forces and moments in the elements are printed. Note that these forces and moments are the forces and moments of the element that act on the nodes. These results can be used to make hand calculations. Another important result of the simulation is the maximum stress in the construction. By clicking on [List Results](#), [Element Solution](#), [Stress](#), [X-component of stress](#) you will see a result such as:

```
PRINT S      ELEMENT SOLUTION PER ELEMENT
```

```
STRESSES AT BEAM SECTION NODAL POINTS
```

```
ELEMENT =      1  SECTION ID =      1
```

```
ELEMENT NODE = 1
```

SEC NODE	SXX	SXZ	SXY
1	377.66	9.9749	-6.6499
3	927.94	-9.9749	-6.6499
9	-373.11	-9.9749	6.6499
7	-923.40	9.9749	6.6499
Max=	927.94	6.6499	9.9749
Min=	-923.40	-6.6499	-9.9749

```
ELEMENT NODE = 2
```

SEC NODE	SXX	SXZ	SXY
1	268.82	9.9749	-6.6499
3	-108.03	-9.9749	-6.6499
9	-264.28	-9.9749	6.6499
7	112.58	9.9749	6.6499
Max=	268.82	6.6499	9.9749
Min=	-264.28	-6.6499	-9.9749

For each element, the internal stresses (normal and shear stress) at both sides of the beam are shown. The identifiers 1,3,9 and 7 stand for the corners of the square cross-section. If you see more values, you should go back to the cross-section and change the values Nh and Nb to 1. You can use this option and the [Plot Results](#) option to find the maximum stresses in the construction.

Finally, with [Reaction Solu](#) the reaction forces in the supports can be printed. These are important in order to check whether equilibrium of external forces and moments are fulfilled. Fill in the questionnaire, based on the results.