Mini SAP 2000 Tutorial for an Example Beam (Hands-on Exercise in the Lab.)
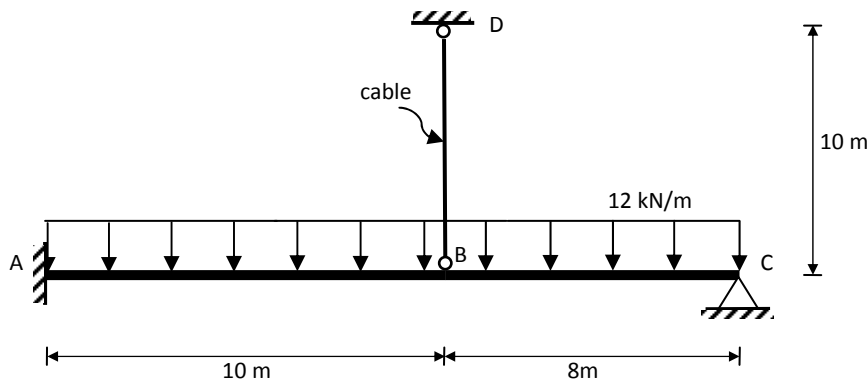

EI_{beam} : $5 \times 10^{4}\ kN.m^2$

EA_{cable} : $4 \times 10^4\ kN$

Beam is axially rigid.

Question: How do we model this beam using SAP2000?

This is a mini-tutorial prepared for the structural analysis of **a sample beam** using **SAP2000**. Please note that the order of steps below is not important, however one should be careful in completing all of them before the analysis. You might need to use other options in other versions or for other problems; this is **only one sample use of the software for a beam analysis**. SAP 2000 has a very strong tutorial itself, which you can Access from the Help menu and you are encouraged to read it whenever you have questions.

1) Define Model Geometry

a) Defining the New Model: The first step when you start SAP2000 is to start a **New Model** under the **File** menu.

b) Basic Units: If you are starting a new model, choose the **Initialize Model from Defaults with Units** you want to use (Fig1). The units used to start a model become the basic units for that model. All material constants and dimensions such as modulus of elasticity, length, area, moment of inertia… should be defined compatible with the basic units. It is possible, however, to use the Units drop-down list on the status bar located at the lower right of the window to change the units at any time.
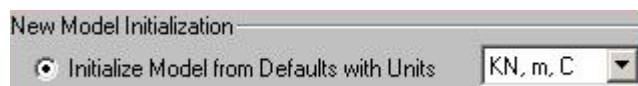

Figure 1

c) Type of Structure: Choosing a template that closely resembles the structure to be analyzed and designed can greatly speed the overall modeling process in most cases. Pick a Structure template from the **Select Template** option (choose beam for our case,Fig2). Please note that there are many other options available according to the type of structure you wish to analyze.

Figure 2

d) <u>Defining Dimensions & Grid Generation:</u> When you select beam, a new window will appear asking the dimensions required to draw the shape of the beam to be analyzed. To edit grids manually tick **Use Custom Grid Spacing and Locate Origin** box and hit **Edit Grid** button (Fig3).
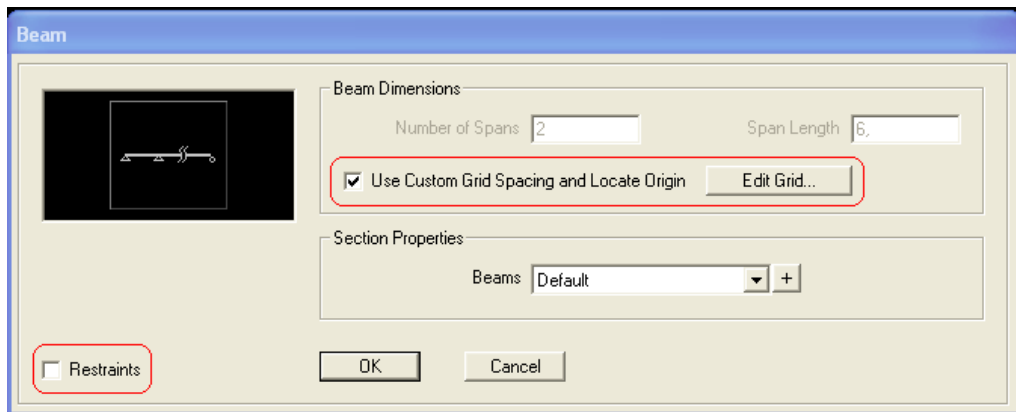

Figure 3

Set ordinates on X Grid Data as x1=0, x2=10, x3=18 and set ordinates on Z Grid Data as z1=0, z2=10 (Fig4). Hit OK after you define ordinates. Do not forget to remove click near the Restraints box to the lower left of beam window(Fig3). Otherwise Sap2000 assigns support conditions automatically.
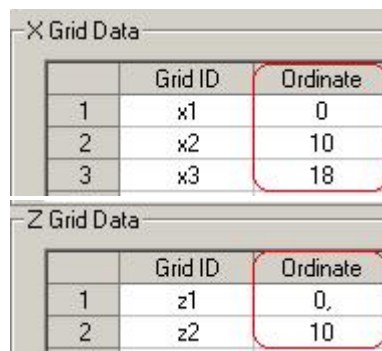

Figure 4

In order to add extra elements to the structure, click **Draw** menu and choose the type of element you wish to add. Choose **Draw Frame/Cable/Tendon** and draw the cable which is connected to the beam.

e) <u>Support Conditions:</u> To add or change the supports conditions, click on the grid point where the support is and choose **Joint** under the **Assign** menu, and select **Restraints** (Fig 5).
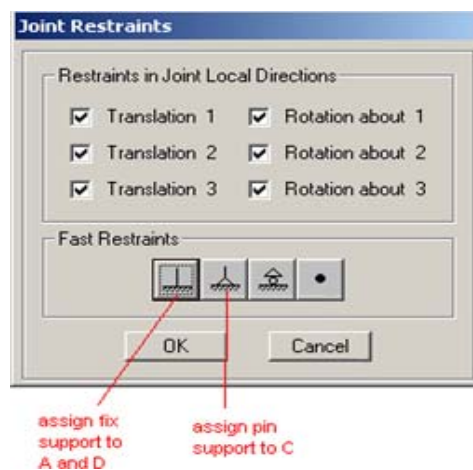

Figure 5

f) <u>End Releases:</u> Cable BD is attached to the beam such that there is no moment transfer between the beam and the cable. In order to model this behavior select cable BD, follow **Assign > Frame > Releases/Partial Fixity** and **Release Moment 33** at start and end joints (Fig 6).

Figure 6

2) Define Material Properties & Frame Sections

a) <u>Material Properties:</u> To assign material properties of the beam and the cable, choose Materials under the Define menu. A window will appear where you can either choose among the default materials or you can define your own material by clicking **Add New Material**. When new material window appears, you can enter properties. (Please note that if you do not want the self-weight of the structure to be considered, enter weight per unit volume as 0.0)

Hint: If flexural rigidity (EI) and axial rigidity (AE) of sections are given take E=1 for simplicity (Fig7).
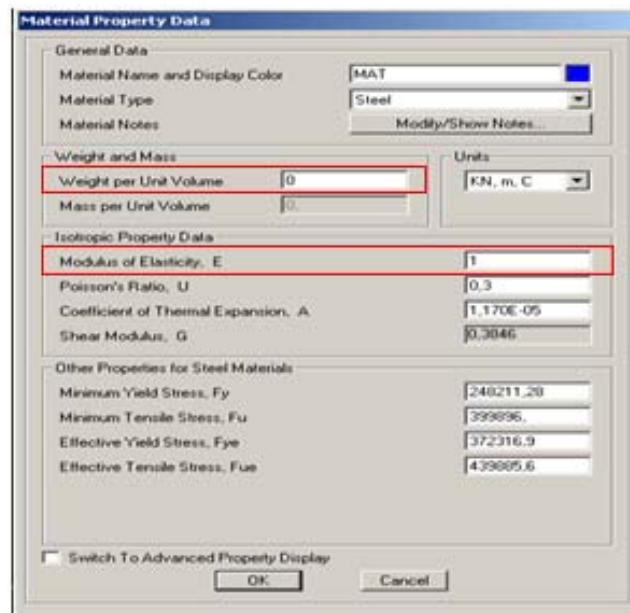

Figure 7

b) <u>Frame Sections:</u> To define the cross-sectional properties of structural elements, click on **Define>Section Properties > Frame Sections**. The **Frame Properties** window will appear. If you do not want to use standard sections, hit **Add New Property** button. **Add Frame Section Property** window will appear, from the **Frame Section Property Type** drop-down menu, choose **Other** and hit the **General** Button (Fig 8). (Please note that one can as well assign a Rectangular frame section and enter the width and depth as well, if the section is rectangular and dimensions are known.)
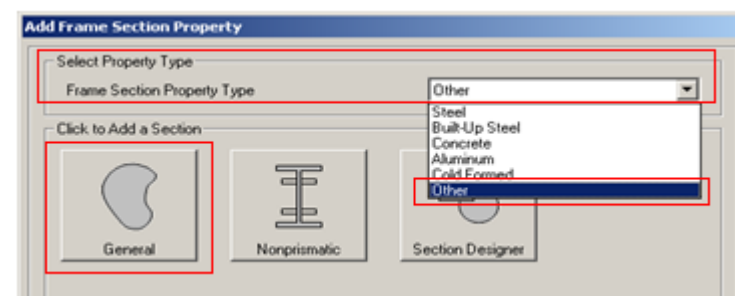

Figure 8

<u>For cable:</u> enter area as $4*10^4$ so that EA=40000kN (E has been assigned as 1). Assign rest of the coefficients as zero to ensure that cable resists to axial deformation only.

<u>For beam:</u> assign a large value to area such as 10e7 in order to provide axial rigidity, EA=∞. Enter moment of inertia about major axis "3" as $5*10^4$ so that EI=50000kN*m^2 (E has been assigned as 1). Assign rest of the coefficients as zero to ensure that beam resists to axial and flexural deformations only.

Do not forget to assign a **Section Name** and the **Material Property** you defined in previous step to your frame section. (Figure 9)
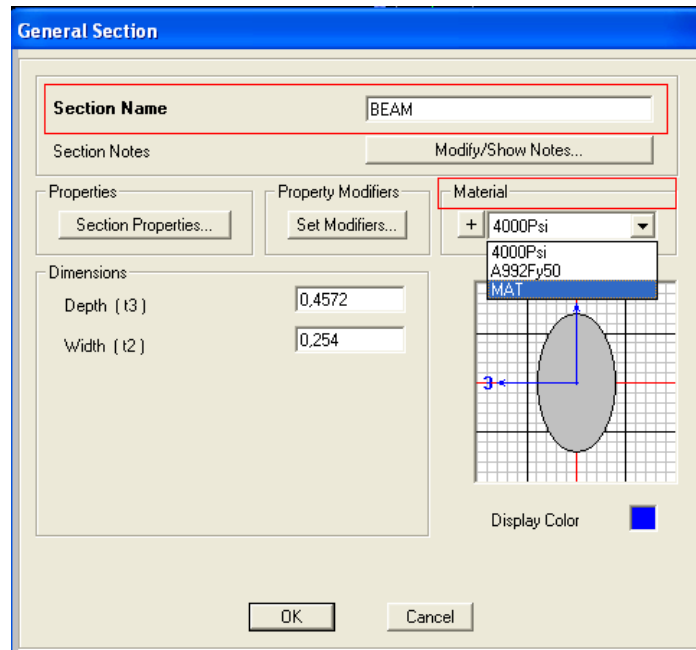

Figure 9

3) Assign Frame Sections & Structural Loads

a) <u>Assigning Frame Sections:</u> In the previous step, you defined two new frame section (cable and beam) where you added the material property you had defined before. Now, you should select the frame members which you wish to assign the frame section properties you defined. In order to assign frame section to a selected member follow: **Assign > Frame > Frame Sections.**

b) <u>Defining Structural Loads:</u> To define distributed loads on the elements, click the member where there is a distributed load, and then follow: **Assign > Frame Loads > Distributed**. (Similarly to define joint forces on the structure, click on the joint where there is a force and select the joint loads from the assign menu.)

4) Analyze

a) <u>Analysis Options:</u> Under the **Analyze menu** choose **Set Analysis Options.** Since we are working in the XZ plane, click on the **Plane Frame** button in the **Fast DOFs** section. As you realized upper section, Available DOFs has been modified according to our choice. Afterwards press **OK.**

b) <u>Run Analysis:</u> Under the **Analyze menu** choose **Run Analysis.** For the case name **Modal** click **Run/Do not Run Case** to prevent modal analysis (modal analysis is not required). Press **Run Now** (Fig 10). You will be asked a file name where the results of the analysis will be saved. (Opening a new folder to save all the output files is recommended.) After you see Analysis done window, click OK.
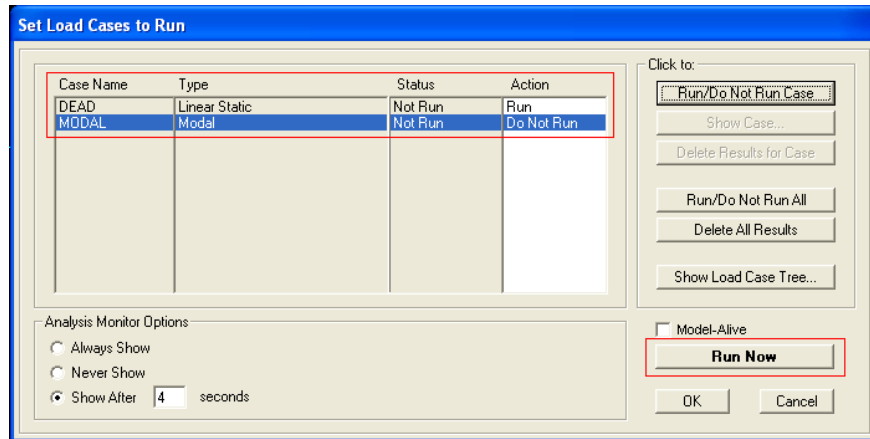
Figure 10

5) Analysis Results

a)Deformed Shape: The default initial output of SAP2000 is the deformed shape of the structure. If you right click on B on this deformed shape, you can see the magnitudes of the deformations at the joint (Fig11).
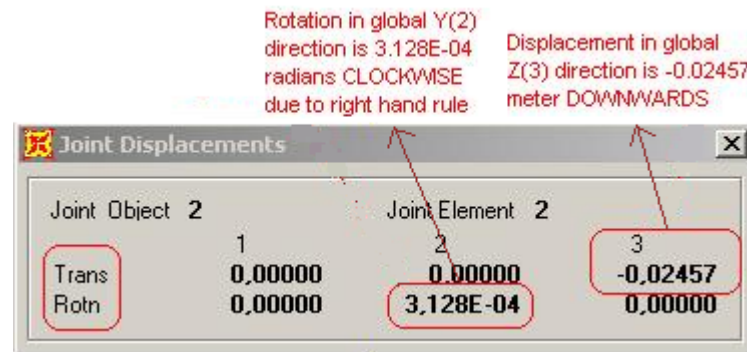

Figure 11

b)Member Forces and Support Reactions: To see the values of the support reactions, or moment, shear and axial force diagrams choose Show Forces/Stresses from the Display menu. Choose Joints or Frames/Cables from the menu. You can see the values of the forces and diagrams you wish to see.

You can also see analysis results in tabular form. Choose **Show Tables** under **Display** menu and select type of result you are interested. You can also export sap2000 analysis results to excel: **Display > Show Tables > tick boxes near interested results > click OK > File > Export Current Table > To Excel.**