

ANSYS TUTORIAL

Analysis of a Simple Cantilevered Beam with End Load

In this tutorial, you will model and analyze the beam below in ANSYS. Step-by-step instructions are provided beginning on the following page.

The steps that will be followed are:

Preprocessing:

1. Change Jobname.
2. Define element type. ("BEAM3", which is a 2-D beam element)
3. Define real constants. (Area, Moment of Inertia, Height, SHEARZ)
4. Define material properties. (Young's Modulus, EX -- only property required for this analysis)
5. Create nodes. (21 total)
6. Create beam elements between nodes. (20 total)

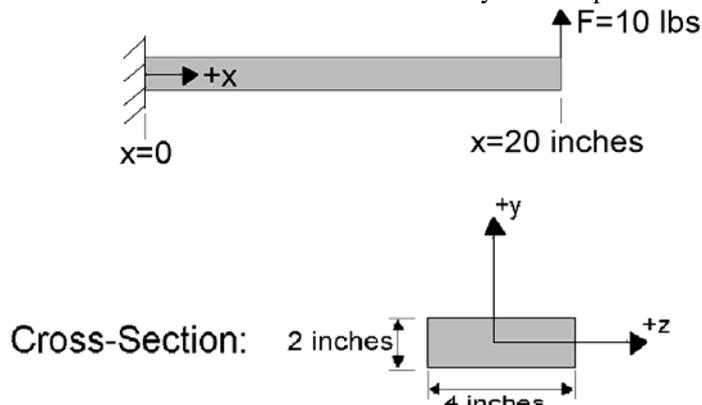
Solution:

7. Apply constraints and loads to the model.
8. Solve.

Postprocessing:

9. Plot deformed shape.
 10. List reactions.
 11. List the deflections at each node.
 12. Define element table items for plotting and listing of various stress components.
 13. List element table items.
 14. Plot element table items.
 15. Exit the ANSYS program.
-

Note: Beam Material Modulus of Elasticity = $30E6$ psi



Note: The instructions below include alternative command line entries that can be ignored if you choose to use menu picks to perform the required tasks. These are provided for your

information. You may find that it is sometimes more convenient to enter certain commands directly at the command line.

*****AS YOU WORK THROUGH THIS TUTORIAL, WITHIN ANSYS, ON THE ANSYS TOOLBAR (UPPER RIGHT) CLICK ON “SAVE_DB” OFTEN!!!**

Preprocessing:

1. Change jobname:

File -> Change Jobname

Enter “beam”, and click on “OK”.

Alternative Command Line Entry = /filnam, beam

Also, to enter the preprocessor, at the command line, enter: /prep7

2. Define element types:

Preprocessor -> Element Type -> Add/Edit/Delete

Click on “Add..”, highlight “Beam”, then “2D elastic 3”, click on “OK”, then “Close”. Note that, in ANSYS, this element is sometimes referred to as “BEAM3”, because it is element type 3 in the ANSYS element library.

Alternative Command Line Entry = et,1,3

3. Define the real constants for the BEAM3 elements:

Preprocessor -> Real Constants -> Add

Click “OK” for “Type 1 BEAM3”

The appropriate values for the given geometry are: AREA=8; IZZ=(32/12); HEIGHT=2; SHEARZ=(6/5). Leave the other fields blank, and click “OK”, then close the “Real Constants” box.

Alternative Command Line Entry = r,1,8,(32/12),2,(6/5)

Note that the “1” following “r,” denotes that this is real constant set number 1. Also, the “SHEARZ” term, entered as (6/5), allows shear deformation effects to be included in the solution. The value used is appropriate for a rectangular cross-section. A different value would be required, for instance, for a circular cross-section. This term can be left blank if shear deformation effects are assumed to be negligible.

4. Define Material Properties:

Preprocessor -> Material Properties -> -Constant- Isotropic

“OK” for material set number 1, then, enter 30E6 for EX, then “OK”.

Alternative Command Line Entry = ex,1,30E6

5. Create nodes:

Preprocessor -> -Modeling- Create -> Nodes -> In Active CS

Enter 1 for node number (ANSYS would automatically number nodes if you leave this blank). Enter the location as (x,y,z)=(0,0,0). Leave the entries for rotation angles blank. (Note: For this problem, all nodes will be on the x-axis, with y=0 and z=0). Click on “Apply”. Define node 21 at (x,y,z)=(20,0,0), then click on “OK” instead of “Apply”.

Automatically fill in the other nodes:

Preprocessor -> -Modeling- Create -> Nodes -> Fill between Nds.

A picking menu will open. Click on node 1 at the origin, then click on node 21 at x=20, then click on “OK” in the picking menu. Accept the defaults on the dialogue box that opens. The defaults are to fill in between nodes 1 and 21 a total of 19 nodes. Click “OK” on that dialogue box, and the equally-spaced nodes will be generated.

Alternative Command Line Entry = n,1,0,0,0 (or, simply: n,1– missing input is interpreted by ANSYS as “zero” in most cases).

Alternative Command Line Entry = n,21,0,0,0

Alternative Command Line Entry = fill,1,21,19

As a check to ensure all nodes were entered correctly, list the nodes:

Utility Menu ->List -> Nodes [OK]

Alternative Command Line Entry = nlist

Turn on node numbering:

Utility Menu -> PlotCtrls -> Numbering.

Check “node numbering”, then click “OK”. The node numbers may already be showing, but this will force the display of node numbers on subsequent plots. At any time, of course, to turn off node numbering, the user can return to the same location, and turn off node numbering.

Alternativ

e Command Line Entry = /pnum,node,1

6. Create beam elements between nodes. One way to do this is the general method shown next. This will be followed by a simpler method.

Preprocessor -> Create -> Elements ->-Auto Numbered-Thru Nodes

A picking menu appears. Pick node 1, then node 2, and click on “APPLY” in the Picking Menu. Then, click on node 2, then node 3, and click on APPLY. Continue until all 20 elements are created. ****IMPORTANT: On the twentieth element, click on “OK”, instead of “APPLY”. If you click on “APPLY”, then on “OK”, you will generate two beam elements between nodes 20 and 21.**

Alternative Command Line Entry = e,1,2

Alternative Command Line Entry = e,2,3 (and so on until all 20 elements are defined).

A simpler method to do this, with just two command line entries, is shown below:

e,1,2
***repeat,20,1,1**

The “*repeat” command is not available through menu picks. This command, as entered above, repeats the previous command a total of 20 times (including the initial time it was input). It increments the values in the two input fields each by 1 each time the command is repeated.

Solution:

If entering commands at the command line, to enter the solution processor, type: /solu

7. Apply constraints and forces on the model:

To apply constraints:

Solution -> -Loads- Apply -> -Structural- Displacement -> On Nodes

Pick node 1, then click “OK” in the picking menu that has appeared. Choose ALL DOF, and use the default value of zero. Click on “OK”.

Alternative Command Line Entry = d,1,all,0

To apply the force, choose:

Solution -> -Loads- Apply -> -Structural- Force/Moment -> On Nodes

Pick node 21, then “OK” in the picking menu, choose “FY” as the direction of the force, and enter “10” for the force value. Click on “OK”.

Alternative Command Line Entry = f,21,fy,10

8. Solve the problem:

Solution -> -Solve- Current LS

Click “OK” in the “Solve Current Load Step” Box.

Alternative Command Line Entry = solve

Postprocessing:

If entering commands at the command line, to enter the postprocessor, type: /post1

9. Plot the deformed shape:

General Postproc -> Plot Results -> Deformed Shape

You will probably want to choose “Def + undeformed”, then “OK”.

Alternative Command Line Entry = pldi,1

10. List reaction forces:

General Postproc -> List Results -> Reaction Solution

Use the default: “All items”; and click on “OK”.

Alternative Command Line Entry = prrf

11. List nodal displacements:

General Postproc -> List Results -> Nodal Solution -> DOF Solution -> ALL DOFs

Alternative Command Line Entry = prdi

12. Define element table items for subsequent plotting and listing of various stress results. For line elements, such as beam elements, to generate certain stress results, you must define an element table, using the ETABLE command. Note that each element is defined by two end nodes, denoted by node i and node j. Node i is the first of the two nodes defined when the element was created. The form of the command to define a table containing the bending stresses at the “top” of the beam, at node i, for each element, is:

ETABLE,BSTRTOPI,LS,2

This command can be entered directly at the ANSYS input line. The entries in this command pertain specifically to the BEAM3 element. If, instead of 2-D BEAM3 elements, the 3-D BEAM4 elements were used, the user would have to look at the on-line help information for BEAM4 to determine the appropriate entries for the ETABLE command.

In the command above, “BSTRTOPI” is simply a label chosen by the user. Practically any other text string could be used in this field. However, it must not have more than eight characters, and it cannot be identical to any general ANSYS predefined label. “LS” is the appropriate item for bending stress in BEAM3 elements (as shown in the on-line help for BEAM3). Also, the “2” shown as the last entry is correct sequence number on the ETABLE command, assuming BEAM3 elements are used, and the user wants to store the bending stresses at the top of the beam elements, at node i for each element.

If the user also wants to define an element table in which the bending stress at the top of each beam element, at node j for each element is stored, the command that could be used is similar to that above. However, the user should choose a different identifying label, and the value of the last entry for this case is “5”. For instance,

ETABLE,BSTRTOPJ,LS,5

The ANSYS on-line help should be consulted for further explanation, and also to view the available stress components and the appropriate ETABLE command entries for storing them. For help on BEAM3 elements, click on “HELP” on the Top Utility Menu, then choose “HELP ON”, enter the number “3”, and click on “OK”. For help on the ETABLE command, click on “HELP” on the Top Utility Menu, then choose “HELP ON”, enter “ETABLE”, and click on “OK”.

As additional examples, to store the bending stress at the bottom of the beam elements, at node i for each element, the command is,

ETABLE,BSTRBOTI,LS,3

and to store the bending stress at the bottom of the beam elements, at node j for each element, the command is,

ETABLE,BSTRBOTJ,LS,6

13. List element table results. Assuming all four of the above ETABLE commands were entered, then a listing of each stored stress component can be obtained with the menu path:

General Postproc -> List Results -> Elem Table Data

Then, highlight the desired labels, and click on “OK”. The stresses corresponding to the chosen labels will be listed by element number.

Alternative Command Line Entry = pretab,BSTRTOPI,BSTRTOPJ

Of course “BSTRTOPI” and “BSTRTOPJ” are the user-defined labels discussed in Step 12, The user can enter any previously defined labels, up to a total of nine labels, on the “pretab” command.

14. Plot element stresses. There are a couple of forms in which the plots can be obtained. One is through use of the “PLLS” command, and the other is through use of the “PLETAB” command. The appearances of the plots will not be described here, but the user can experiment using the following approach. This discussion assumes the user has defined the same element tables, and used the same labels, as overviewed in Step 12.

----Stress plots using the “plls” option----

General Postproc -> Plot Results -> Line Elem Res

In the dialogue box that opens, the user could select the label “BSTRTOPI” for the LABI entry, and the label “BSTRTOPJ” for the LABJ entry. The optional scale factor can be left as “1” if desired, and the user can choose whether or not to include on the plot either “Deformed Shape Only”, “Deformed + Undeformed”, or “Deformed + Underormed Edge”.

Alternative Command Line Entry = plls,BSTRTOPI,BSTRTOPJ

Of course, the other component labels could be chosen instead, if desired.

-----Stress plots using the “pletab” option-----

General Postproc -> Plot Results -> Elem Table

In the dialogue box that opens, the user could select the label “BSTRTOPI”, or of course, any of the other previously defined lables could be chosen. Also, the user has the option of averaging nodal values (see on-line help for a discussion of the averaging).

Alternative Command Line Entry = pletab,BSTRTOPI

Where, again, the other components could be chosen instead, if desired.

15. Exit ANSYS. Toolbar: *Quit ->Save Everything -> OK*