AN INTRODUCTORY ANSYS TUTORIAL: SOLVING A STATIC TRUSS PROBLEM

Rajesh Bhaskaran
Cornell University
E-mail: rb88@cornell.edu

This is a quick-and-dirty introductory tutorial to the ANSYS software package that details how to solve a simple static truss problem. ANSYS is a finite-element analysis package used widely in the industry to simulate the response of a physical system to structural loading, and thermal and electromagnetic effects. ANSYS uses the finite-element method to solve the underlying governing equations and the associated problem-specific boundary conditions. No previous knowledge of ANSYS is assumed in this tutorial.

As you go through the tutorial, mark suggested changes and problems, and hand in a description of what you learned. The tutorial is followed by two problem sets. Record and submit how much time you spent on the tutorial and each of the problem sets.

Problem Specification

Determine the force in each member of the following truss. Indicate if the member is in tension or compression. The cross-sectional area of each member is 0.01 m^2 and the Young's modulus is 200×10^9 N/m^2.

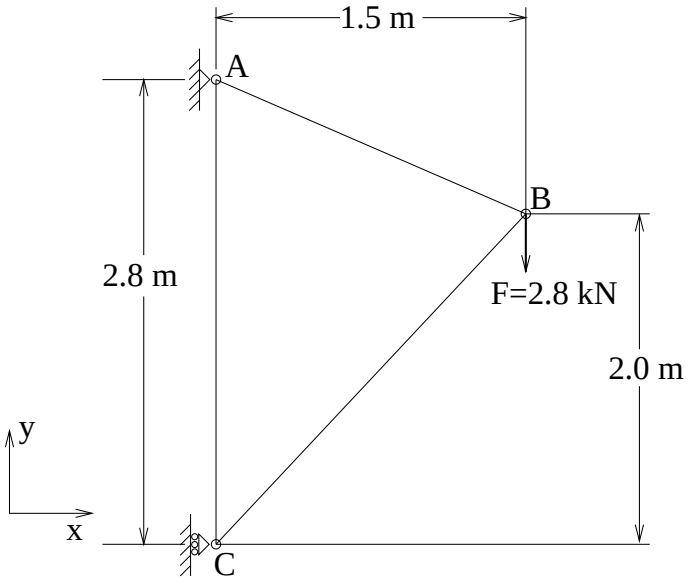


Figure 1: Geometry of the truss.

Solution Steps

1. Start-up and preliminary set-up
2. Specify element type and constants

3. Specify material properties
4. Create nodes
5. Create elements from nodes
6. Specify boundary conditions
7. Solve!
8. Analyze the solution

Let's go through these steps now.

Start-Up and Preliminary Set-Up

ANSYS is available at the ACCEL computer lab on the second floor of Carpenter Hall. To get to ACCEL, go past the circulation desk of the Engineering library and take the stairs to your left. Log into Windows at the ACCEL facility. If you don't have an account, follow the instructions on the screen to create an account. ANSYS needs a directory into which it can save files during the session. Start *NT Explorer* from the desktop and create a folder called *ansys* on the *S:* drive. Minimize the *NT Explorer* window. At ACCEL, the *S:* drive puts you in your home directory which is saved from session to session and can be accessed from any machine at ACCEL.

To start ANSYS, click on

Start → *Programs* → *Ansys 5.5* → *Interactive*

In the window that pops up, enter *S:\ansys* as your working directory. This is the directory to which ANSYS will write files. Specify *truss* as your *Initial jobname*. The jobname is the prefix used for all files generated by the ANSYS run. Click on *Run*. This brings up the following windows:

- ANSYS/University High Option Utility Menu: This is referred to simply as the Utility Menu. Note that the jobname *truss* appears in parenthesis in the title bar of the Utility Menu.
- ANSYS Main Menu
- ANSYS Graphics
- ANSYS Toolbar: This contains shortcuts to often-used commands and can be customized.
- ANSYS 5.5 Output Window (which might be hidden behind the others): This is the window to which output is written by default and provides feedback on the actions taken by ANSYS as you navigate the menus.

We'll more or less work our way down the Main Menu. In the Main Menu, click on the *Preferences* button. In the *Preferences for GUI Filtering* dialog box, click on the box next to *Structural* so that a tick mark appears in the box. The effect of selecting the structural preference is that only menu options applicable to structural mechanics are shown in the graphical user interface; options related to other disciplines such as thermal, fluid and electromagnetics are grayed out. This helps a little while navigating the vast menu options within ANSYS. Click on *OK* to close the *Preferences* dialog box.

Specify Element Type and Constants

In the Main Menu, click on *Preprocessor* to enter the preprocessor module in ANSYS. This is the module where you create the geometry, specify appropriate displacement constraints and loading, and mesh the geometry. We are more or less going to march down this menu to set up the problem. This type of menu is referred to as a "sticky menu" since it sticks around until you explicitly close it. You will find yourself negotiating through a series of sticky menus as you work off the Main Menu.

To specify the element type, click on

Element Type → *Add/Edit/Delete...* → *Add...*

Pick *Structural Link* in the left field and *2D spar 1* in the right field. Click *Apply* to select this element. Note that *LINK1* appears as the only defined element type in the *Element Types* dialog box. To view the help pages for this

element type, click on *Help* in the *Library of Element Types* dialog box and then, in the box that comes up, select

Pick Elements 3.2: Pictorial Summary

and click *Go To*. This will bring up the the help page containing the pictorial summary of the element types available in ANSYS. Our own humble *LINK1* element is listed here. The "1" in the element name refers to the reference number of this element type in ANSYS's list of available element types. Note that the degrees-of-freedom (DOFs) for the *LINK1* element are UX and UY i.e. the displacements in X and Y directions, respectively. It's the simplest way you can link two nodes. Note that in general you need to take the time to understand the element types and pick the appropriate one(s) for your problem. This choice has a significant effect on the speed and accuracy of the solution. Minimize the help window. Click *Cancel* to close the *Article Selection* dialog box. Click *Cancel* to close the *Library of Element Types* dialog box. Now you will see the following in the *Element Types* dialog box:

```
Defined Element Types:
Type 1 LINK1
```

Click *Close* and also close the *Element Type* sticky menu by clicking on the "x" in the right corner of the title bar.

Specify Element Constants

Next we'll specify the constants for this element type. In the *Preprocessor* menu, click on

Real constants $\rightarrow$ *Add...*

This opens up the *Element Type for Real Constants* dialog box. We have only one element type defined from the previous step and it's automatically selected. Click *OK*. For *AREA*, enter *0.01* which is the cross-sectional area of the element. Leave the *Initial Strain* field blank since it's not applicable to our problem. Click on *OK*. Click on *Close* in the *Real Constants* dialog box.

Specify Material Properties

In the *Preprocessor* menu, click on *Material Props* and then on *Isotropic*. Click on *OK* in the *Isotropic Material Properties* dialog box to accept the default material number of 1 (this is just a reference number for book-keeping). Enter 200e9 for Young's modulus *EX* and click *OK*. Close the *Material Props* sticky menu.

Save your work by clicking on the *SAVE_DB* button in the ANSYS Toolbar or equivalently, by clicking on *File* $\rightarrow$ *Save as Jobname.db* in the Utility menu. This saves all the relevant data into one file called "truss.db" in your working directory, "truss" being taken from the jobname and "db" being an abbreviation for database. Verify that ANSYS has created such a "database file" in your working directory (*S:\ansys*). You can restart from your last save at any time by clicking on *RESUME_DB* in the toolbar or by using *File* $\rightarrow$ *Resume Jobname.db* in the Utility menu. Each time you successfully finish a series of steps, you should save your work. Unfortunately, ANSYS doesn't have an undo button (though that is the first thing I needed while learning ANSYS!) and one way to recover from mistakes is to resume from your last save.

Create Nodes

Usually, one creates the geometry and then uses ANSYS meshing routines to create the nodes and elements. In our case, the geometry is simple and we will instead create the nodes and elements manually. Each truss member can be represented by a single *LINK1* element with a node at either end. So we'll need three nodes, located at *A, B* and *C* in figure 1. We'll locate the origin of the coordinate system at *C* and number the nodes at *A, B* and *C* as 1, 2 and 3, respectively.

To create the nodes, in the *Preprocessor* menu, click on

-Modeling- Create $\rightarrow$ *Nodes* $\rightarrow$ *In Active CS ...*

The active CS (i.e. Coordinate System) is the global Cartesian system by default and we'll work only in this coordinate system in our friendly introduction. ANSYS offers the capability to switch between various types of coordinate systems which will be necessary when you move on to solving super-duper problems.

In the *Create Nodes in Active Coordinate System* dialog box,

Enter 1 for node number

Enter 0 for X and 2.8 for Y (The rest default to zero)

Click *Apply* (which accepts the input and then brings back the dialog box for further input).

Enter 2 for node number
Enter 1.5 for X and 2.0 for Y
Click *Apply*
Enter 3 for node number
Enter 0 for X and 0 for Y
Click *OK* (which accepts the input and then closes the dialog box; note the difference between *Apply* and *OK* which holds throughout ANSYS).

The nodes will now be displayed in the Graphics window along with a triad that indicates the origin of the coordinate system (coincident with node 3 in our case) and the axes. To check if the nodes have been created correctly, go to the Utility Menu (usually in the topmost part of the screen) and click on *List* → *Nodes* . This brings up the *Sort NODE Listing* dialog box. Click *OK*. This brings up a window listing the coordinates and rotation angles for the nodes. Verify that you have the following:

NODE	X	Y	Z	THXY	THYZ	THZX
1	0.0000	2.8000	0.0000	0.00	0.00	0.00
2	1.5000	2.0000	0.0000	0.00	0.00	0.00
3	0.0000	0.0000	0.0000	0.00	0.00	0.00

You can rotate the coordinate system associated with each node and that is what the rotation angles *THXY, THYZ* and *THZX* refer to. In our case, we don't need to rotate the nodal coordinate system and so the rotation angles are identically zero. Close the window listing the nodes. Close the *Nodes* and *Create* sticky menus.

If you are like me, you made a mistake while creating nodes and cursed that there is no undo button. To correct your mistake(s), you can delete nodes and re-create them. In case you need to delete a node, in the *Preprocessor* menu, click on *Delete* → *Nodes*. This brings up the so-called pick menu. Click on the node you want to delete; then click on *OK* in the pick menu. You should see the node disappear in the Graphics window. You can also check that the node has been deleted using *List* → *Nodes* . You can then re-create the node.

Once you have successfully created the nodes, click on *SAVE_DB* in the toolbar to save the database.

Create Elements From Nodes

Recall that we are representing each truss member as a single *LINK1* 2D spar element. To create elements from the nodes, in the *Preprocessor* menu, click on

Create → *Elements* → *-Auto Numbered- Thru Nodes*

This will bring up the *Elements from Nodes* pick menu. First, create the element between nodes 1 and 3 as follows:

- In the Graphics window, click on node 1.
  ANSYS will draw a little square around node 1 to indicate that it's been "picked".
- Then, click on node 3 (at the bottom of the screen overlapping the triad).
- Click *Apply* in the *Elements from Nodes* pick menu.
  This will draw a line between nodes 1 and 3 which denotes the *LINK1* 2D spar element between these nodes.

Next, create the element between nodes 1 and 2 as follows:

- In the Graphics window, click on node 1.
- Click on node 2.
- Click *Apply* in the pick menu.
  This will draw a line between nodes 1 and 2 which denotes the *LINK1* element between these nodes.

Last, create the element between nodes 2 and 3:

- In the Graphics window, click on node 2.
- Click on node 3.

- Click *OK* in the pick menu. (We clicked on *OK* instead of *Apply* since this is the last element to be created and *OK* accepts the input and then gets rid of the menu.)

You can now get rid of the *Elements* and *Create* sticky menus.

To take a look at the list of elements, in the Utility Menu, click on

List → Elements → Nodes + Attributes

I get the following list in the window that comes up:

```
  ELEM MAT TYP REL ESY SEC        NODES

     1   1   1   1   0   1       1     3
     2   1   1   1   0   1       1     2
     3   1   1   1   0   1       3     2
```

This table says that Element 1 is of material type 1 and element type 1 and is attached to nodes 1 and 3 and so on. In this element list, the order of the two nodes for each element doesn't matter. For example, element 3 can be attached to nodes 2 and 3 or equivalently, nodes 3 and 2. If one or more elements doesn't look right, you can delete it and re-create that element. To delete an element, in the *Preprocessor* menu, click on *Delete* → *Elements*. This brings up the pick menu. Click on the element you want to delete in the Graphics window and click *OK*. This deletes the element.

Close the window listing the elements. Once you have successfully created the elements, click on *SAVE_DB* in the toolbar to save the database.

Specify Boundary Conditions

Apply Displacement Constraints

Next, we step up to the plate to define the boundary conditions, namely, the displacement constraints and loads. Note that in ANSYS terminology, the displacement constraints are also "loads". From figure 1, the pin at *A* is constrained in *x* and *y* directions; or equivalently, node 1 is constrained such that its *UX* and *UY* displacements are zero. Similarly, node 3 is constrained such that its *UX* displacement is zero.

To define the displacement constraints, in the *Preprocessor* menu, click on

Loads → *-Loads- Apply* → *Displacement* → *On Nodes*

This brings up the *Apply U,ROT on Nodes* pick menu. Click on node 1 in the Graphics window; this will draw a small square around node 1 to indicate that it's been picked. Click *Apply* in the pick menu. Select both *UX* and *UY* from items in *DOFs to be constrained* list. Since the *Displacement value* is zero by default, leave that field empty. Click on *Apply*. You will see triangle symbols appear in the Graphics window indicating that both UX and UY DOFs are constrained at node 1.

Next, click on node 3 in the Graphics window and click on *OK* in the pick menu. Select only *UX* from items in *DOFs to be constrained* menu. Click *OK*. You will see a triangle symbol appear indicating that only the UX DOF is constrained at node 3. Close the *Displacement* and *Apply* sticky menus.

List the displacement constraints on the model by clicking on

List → *Loads* → *DOF constraints* → *On All Nodes*

in the Utility Menu. This brings up a window with the constraint information:

```
  NODE  LABEL     REAL            IMAG
     1  UX      0.00000000      0.00000000
     1  UY      0.00000000      0.00000000
     3  UX      0.00000000      0.00000000
```

Close this window. Click on *SAVE_DB* in the toolbar to save the database.

Apply Loading

We have only one load to apply on the structure, namely, the 2800 N force in the negative *y*-direction at node 2. In the *Loads* sticky menu, click on

-Loads- Apply → *Force/Moment* → *Nodes*

This brings up the pick menu. Click on node 2 in the Graphics window; then click *OK* in the pick menu. In the *Apply*

F/M on Nodes dialog box that appears, select *FY* for *Direction of force/mom*. Enter -2800 for *Force/moment value*. Click *OK*. The negative sign for the force indicates that it is in the negative *y*-direction. You'll see a vector indicating the applied force in the Graphics window.

Click on *SAVE_DB* in the toolbar to save the database. Close the *Force/Moment*, *Apply*, *Loads* and *Preprocessor* menus.

Solution

Enter the solution module in ANSYS by clicking on *Solution* in the Main Menu. Under *Solve*, click on *Current LS*. This solves the current load step (LS) i.e. the current loading conditions. In our problem, there is only one load step; ANSYS allows for multiple load steps that can be solved sequentially without leaving the *Solution* module. Review the information in the */STAT Command* window which is a summary of the problem that ANSYS is about to solve. Close this window. Click *OK* in *Solve Current Load Step* dialog box. ANSYS performs the solution and a yellow window should pop up saying *Solution is done!*. Congratulations! You just obtained your first ANSYS solution. Close the yellow window.

Click on *SAVE_DB* in the toolbar to save the database. In preparation for the post-processing step to be undertaken next, exit the solution module by closing the *Solution* sticky menu.

Postprocessing

To analyze the solution obtained, enter the *General Postprocessing* module by clicking on *General Postproc* in the Main Menu. To list the forces in the members, click on *List Results* $\rightarrow$ *Element Solution*. Select *Nodal force data* from list on left. Select *All forces FORC* from list on right. Click *OK*. This brings up a window listing the forces that the elements apply on each of their nodes:

```
 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
       1   0.0000     -2000.0
       3   0.0000      2000.0

  ELEM=       2  FX          FY
       1   1500.0     -800.00
       2  -1500.0      800.00

  ELEM=       3  FX          FY
       2   1500.0      2000.0
       3  -1500.0     -2000.0
```

For example, Element 2 (or member AB) applies a force of 1500 N in the x-direction and 800 N in the negative y-direction on node 1 (or pin A). This means that the total force in AB is $\sqrt{1500^2 + 800^2} = 1700\ N$. The resultant acts from A to B i.e. the member is pulling on pin A. So it must be in tension. Similarly, the force in Element 1 (AC) is 2000 N (tension) and in Element 3 (BC) is 2500 N (compression).

To list the reaction forces at the nodes, click on *List Results* $\rightarrow$ *Reaction Solu* . Select *All struc forc F* for *Item to be listed* and click *OK*. This brings up a window with the reaction forces at the nodes:

```
 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
     1  -1500.0      2800.0
     3   1500.0

TOTAL VALUES
VALUE    0.0000      2800.0
```

The sum of the reaction forces balances the applied load as should be the case for static equilibrium.

To take a look at the deformed shape of the structure, in the Main Menu, click on *General Postproc* → *Plot Results* → *Deformed Shape*. Select *Def + undef edge* and click on *OK*. This plots the deformed and undeformed shapes in the Graphics window. Note that the deformation is magnified in the plot so as to be easily visible.

To exit ANSYS, in the Utility Menu, click on *File* → *Exit*. Select *Save Everything* and click *OK*. You can log out from your account by typing *Ctrl+Alt+Delete*.

This is just a quick introduction to ANSYS to give you a flavor of what a full-fledged engineering package looks like. If it felt unfriendly or cumbersome, you are not alone; I went through this myself (otherwise, congratulations! you are a genius). It takes some getting used to. Believe it or not, it gets a lot easier to use with time. You have a lot of years ahead of you to gain the experience necessary to harness the power of finite-element analysis. All the ANSYS features including the underlying theory are documented online and can be accessed from the *Help* button in the Utility Menu. That's the best way to learn more about the program. There are tutorials available in the documentation which is a good place to start. In the latest version, ANSYS 5.7, the documentation is HTML-based and so is similar to navigating web pages. We will have the 5.7 version installed at ACCEL in the fall.

Problem Set 1

Resume the ANSYS tutorial and make modifications to it as discussed below. To resume the ANSYS tutorial, start ANSYS; specify the same directory as in the tutorial but use a different jobname. Once ANSYS comes up, in the Utility Menu, click on *File* → *Resume from*. Choose *truss.db* and click *OK*.

Consider the case where the displacement constraints at A and C are interchanged i.e.

- at A, only UX is set to zero,
- at C, both UX and UY are set to zero.

Before you solve the problem in ANSYS, think about the following:

1. How would you expect the reaction forces at the supports A and C to change?
2. What can you say about how the x-component of the forces in the truss members will change?

Re-solve the truss problem with the modified constraints. You can delete constraints using

Preprocessor → *Loads* → *-Loads-Delete*

It works similar to how you apply loads.

Tip: You can plot the displacement constraints in the Graphics window as follows: in the Utility Menu, click on *Pltctrls* → *Symbols*

Select *All Applied BCs* for *Boundary condition symbol*. Click on *OK*. You might have to use *Plot* → *Replot* or *Plot* → *Multi-Plots* in the Utility Menu for the constraint symbols to appear in your plot.

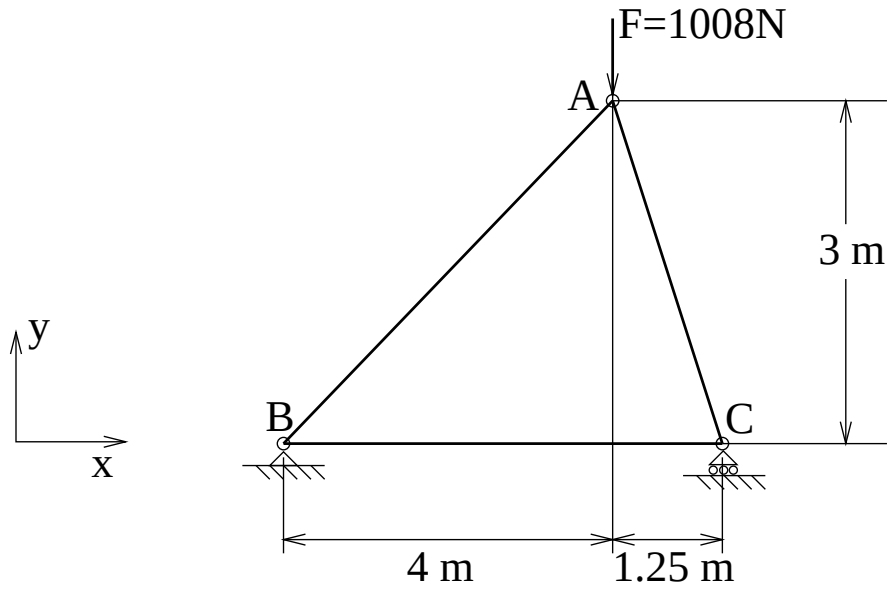
Submit the following results from your ANSYS solution:

1. Listing of the reactions. Note that you can save the reaction listing as follows: in the window that comes up with the listing of the reaction forces, click on *File* $\rightarrow$ *Save as*.

2. Listing of element forces. From this, determine the force in each member and whether the member is in tension or compression.

Problem Set 2

Determine the force in each member of the following truss using ANSYS. Indicate if the member is in tension or compression. Use the same LINK1 element as in the tutorial. The cross-sectional area of each member is 0.02 m^2 and the Young's modulus is 200×10^9 N/m^2. Verify your results by calculating the forces manually.



Submit the following:

1. Listing of the reactions from the ANSYS solution. Indicate which node corresponds to which point (i.e. A,B or C) in the figure.

2. Listing of the element forces from the ANSYS solution. Indicate which element corresponds to which truss member.

3. Your calculation determining the force in each member and whether the member is in tension or compression from the ANSYS result.

4. Your pencil-and-paper calculations verifying the ANSYS results for the member and reaction forces.