ANSYS - 2-D Static Truss

Unknown macro: {rate}

- Problem Specification
- Step 1: Start-up & preliminary set-up
  - Start ANSYS
  - Set Preferences
- Step 2: Specify element type and constants
  - Enter the Preprocessor module
  - Specify Element Type
  - Specify Element Constants
- Step 3: Specify material properties
  - Save your work
- Step 4: Specify geometry
  - Overview
  - Create Keypoints
  - Check Keypoints
  - Save your work
  - Create Lines from Keypoints
  - Check Lines
  - Modifying Lines (If Necessary)
  - Save your work
- Step 5: Mesh geometry
  - Set Meshing Parameters
  - Set Mesh Size
  - Mesh Lines
  - View Element List
  - View Node Location
  - Save your work
- Step 6: Specify boundary conditions
  - Apply Displacement Constraints
  - List Displacement Constraints
  - Apply Loading
- Step 7: Solve!
  - Enter Solution Module
  - Review the Problem
  - Perform Solution
- Step 8: Postprocess the results
  - Plot Deformed Shape
  - Turn On Node and Element Numbers
  - List Forces in Truss Members
  - List Reaction Forces at Nodes
- Step 9: Validate the results
  - Simple Checks
  - Refine Mesh
  - Compare with Theory
  - Exit ANSYS
- Problem Set 1
- Problem Set 2
  - Results

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results
Problem Set 1
Problem Set 2

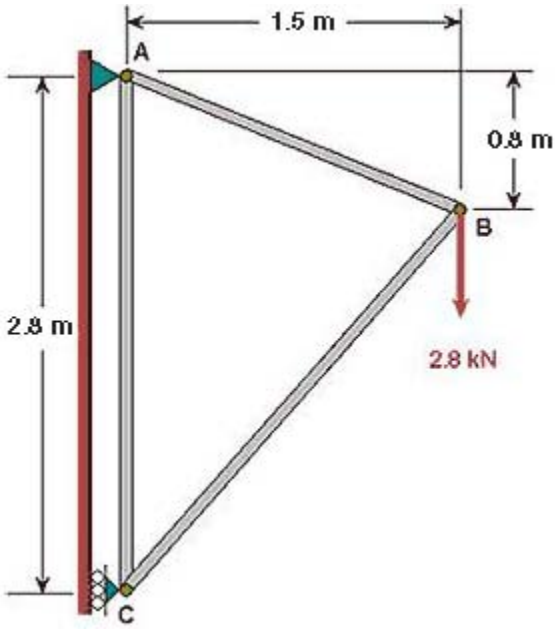
This is an introductory tutorial on the ANSYS software package that details how to solve a simple static truss problem. No previous knowledge of ANSYS is assumed in this tutorial.

Author: Rajesh Bhaskaran, Cornell University

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results
Problem Set 1
Problem Set 2

Problem Specification

Determine the force in each member of the following truss using ANSYS 12.1. Indicate if the member is in tension or compression. The cross-sectional area of each member is 0.01 m, the Young's modulus is 200×10^9 N/m^2 and Poisson ratio is 0.3.



The solution in ANSYS for this and subsequent tutorials is divided into the nine steps listed in the grey area above. Note that you'll need to follow these same nine steps for solving almost any problem in ANSYS or a comparable finite-element analysis package.

Go to Step 1: Start-up and preliminary set-up

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results
Problem Set 1
Problem Set 2

Step 1: Start-up & preliminary set-up

Take a few minutes to review the conventions used in the tutorials which are given on the main page.

Start ANSYS

Create a folder called *ansys* at a convenient location. We'll use this folder to store files created during the session.

There are two ways to launch ANSYS.

1. Using the ***ANSYS Product Launcher*** is the elegant way to start ANSYS. One specifies the working folder, job name etc. in the *Product Launcher*. We will use this approach below.
2. One can alternatively start the ANSYS graphical user interface (GUI) directly and then set the jobname etc. within the GUI. This can be messy with the beginner often not sure where the files from the session are being stored. This is not the recommended way to start ANSYS.

Start the *ANSYS Product Launcher* from the Start menu. The usual location is:

Start > All Programs > ANSYS 12.0 > Mechanical APDL Product Launcher

Note that in version 11, it is at

Start > Programs > ANSYS 11.0 > ANSYS Product Launcher

In the window that comes up, enter the location of the folder you just created as your ***Working Directory*** by browsing to it. If you specify, say, *C:\ansys* as your working directory, all files generated during the ANSYS run will be stored in this folder.

Specify `truss` as your ***Job Name***. The job name is the prefix used for all files generated during the ANSYS run. For example, when you perform a save operation in ANSYS, it'll store your work in a file called *truss.db* in your working directory.

Click on ***Run***. This brings up the ANSYS interface. To make the best use of screen real estate, move the windows around and resize them so that you approximate this screen arrangement. This way you can read instructions in the browser window and implement them in ANSYS. Note that this tutorial has been formatted to fit in a skinny browser window. If your monitor screen is small, you can use *Alt+Tab* keys to conveniently switch between the ANSYS and browser windows (this trick works in Microsoft Windows).

You can resize the text in the browser window to your taste and comfort as follows:

In Internet Explorer: **Menubar > View > Text Size**, then choose the appropriate font size.

In Mozilla Firefox: **Menubar > View > Zoom**

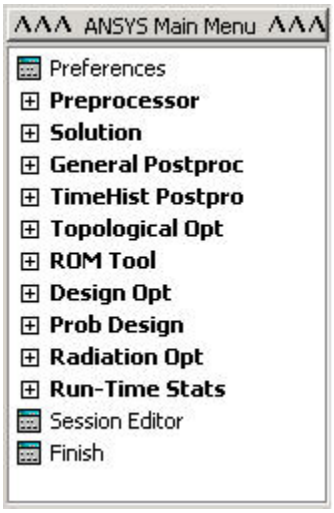
The ANSYS interface consists of the following entities:

- ***Utility Menu*:**



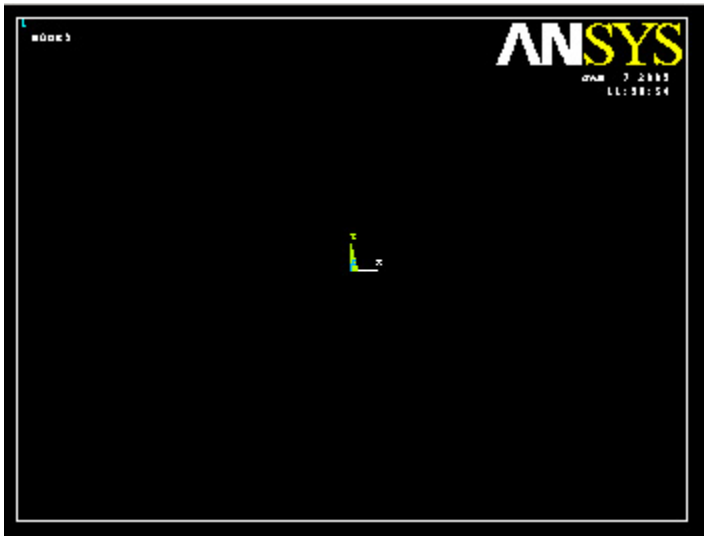
Note that the job name *truss* appears in parenthesis in the title bar of the Utility Menu.

- ***ANSYS Main Menu*:**



We'll more or less work our way down the ***Main Menu*** as we go through the solution steps.

- ***ANSYS Graphics:***



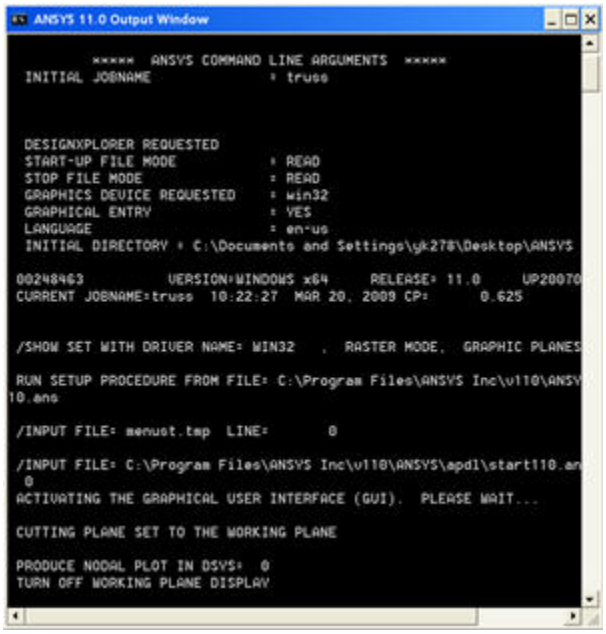
This is the window where the geometry, mesh, applied boundary conditions, and results such as stress values are displayed.

- ***ANSYS Toolbar:***



ANSYS Toolbar contains shortcuts to often used commands such as *save* and can be customized by the user for convenience.

- ***ANSYS 11.0 Output Window:***



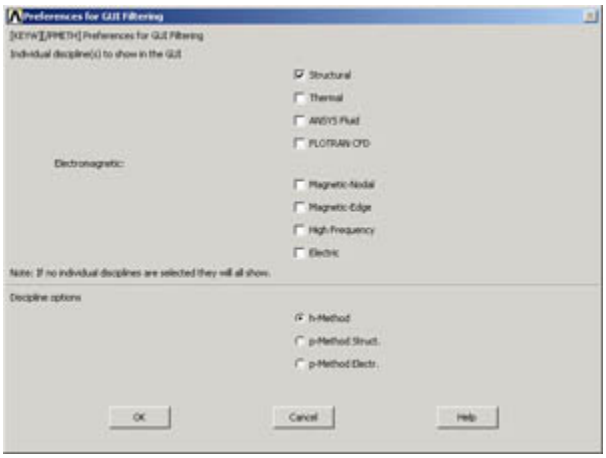
This is the window to which output from ANSYS commands is written and which provides feedback on the actions taken by ANSYS as you navigate the menus. If, at some point, you are not sure you clicked the right button or entered a value correctly, this is where to look first to figure out what you just did.

Set Preferences

We'll more or less work our way down the ***Main Menu***.

Main Menu > Preferences

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 ***OK*** to close the ***Preferences*** dialog box.

Go to Step 2: Specify element type and constants

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results
Problem Set 1
Problem Set 2

Step 2: Specify element type and constants

Enter the Preprocessor module

Main Menu > Preprocessor

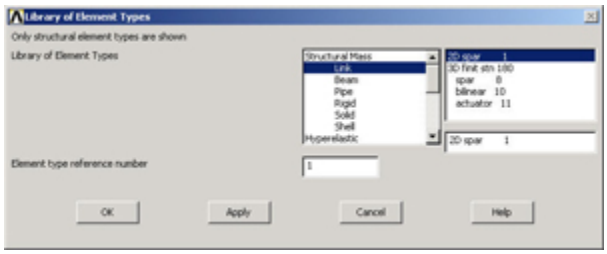
This is the module where you create the geometry, specify appropriate displacement constraints and loading, and mesh the geometry. We are more or less going to march down this menu to set up the problem. You will find yourself negotiating through a series of menus as you work off the ***Main Menu***.

Specify Element Type

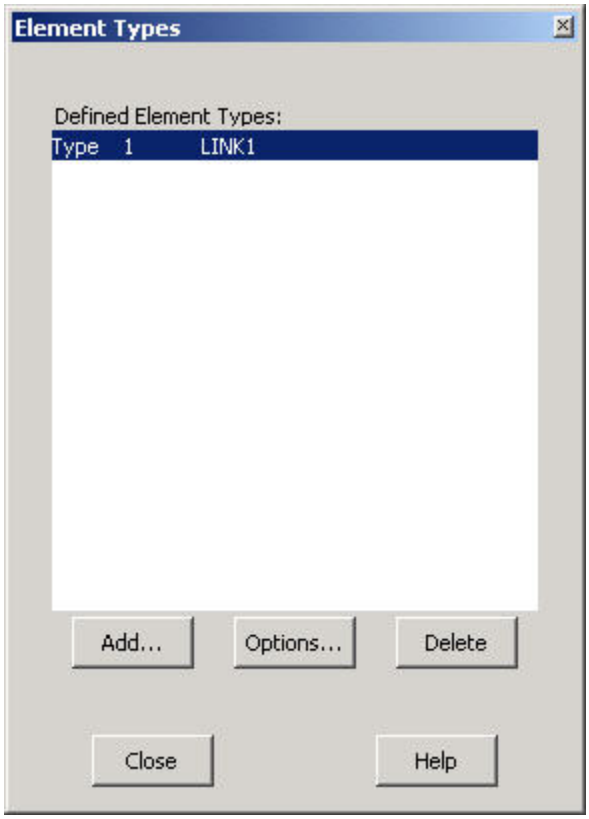
ANSYS provides a large selection of element types to solve linear and non-linear problems in structural mechanics, heat transfer, fluid mechanics and electro-magnetics. We next select the appropriate element type for our problem from this large list. Consider this as equivalent to rifling through a sizable chest, picking out one or more entities and placing them on a table for later use (in step 5, in our case). Select

Main Menu > Preprocessor> Element Type > Add/Edit/Delete > Add...

Pick ***Structural Link*** in the left field and ***2D spar 1*** in the right field. Click **OK** to select this element.



Now you will see the following in the Element Types dialog box:



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 ***Element Types*** dialog box. This brings up the Help window. Click on ***Search*** in the left pane and type in LINK1. (If the left pane is hidden, click the Show button in the toolbar). The first search result is the help page for *the LINK1* element. Bring up this help page by double-clicking on the search result. Note that this is a two-dimensional spar element that supports uniaxial tension and compression but not bending, so it is appropriate for modeling a truss structure. There are two degrees of freedom at each of its two nodes: translations in the nodal x and y directions. The "1" in the element name is the internal reference number for this element type in ANSYS' list of available element types.

Before proceeding, let's take a quick peek at the pictorial summary of the element types available in ANSYS. Search for "pictorial summary" and double-click on the search result titled *3.2 Pictorial Summary*. Click on the link to *Link Elements*. You will see our own humble *LINK1* element as well as other link elements in the pictorial summary. Clicking on the *LINK1* link will take you to the help page for the element that we just visited. In general, you need to take the time to understand the element types and pick the appropriate one(s) for your problem. The pictorial summary is a good place to start for identifying the appropriate element type for your problem. Your choice of element type has a significant effect on the speed and accuracy of the solution. Quickly browse through the pictorial summary to get an idea of the wide variety of element types available in ANSYS. This is what allows the software to solve engineering problems from a wide variety of disciplines.

Minimize the Help window.

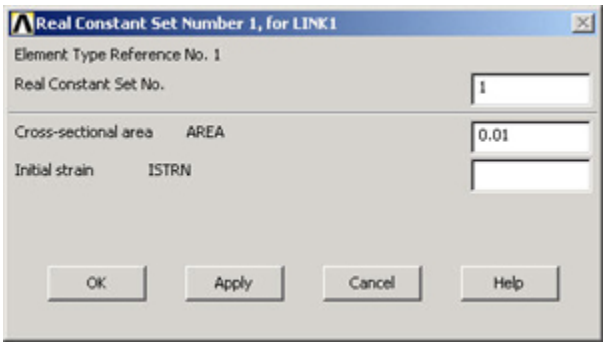
Close the *Element Types* window by clicking ***Close***.

Specify Element Constants

Main Menu > Preprocessor > Real Constants > Add/ Edit/ Delete

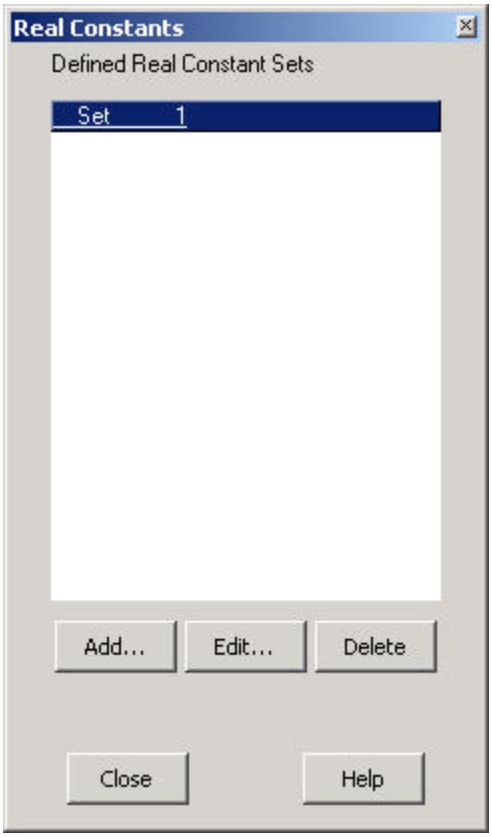
This opens up the *Real Constants* dialog box. Click ***Add...*** This brings up the *Element Type for Real Constants* dialog box with a list of the element types defined in the previous step. We have only one element type, *LINK1*, defined and it's automatically selected. Click ***OK.***

We now enter the constants needed for the *LINK1* element. For ***AREA***, enter `0.01` which is the ***Cross-sectional area*** of the element. We'll work in SI units. Leave the ***Initial strain*** field blank since it's not applicable to our problem.



It is the responsibility of the ANSYS user to make sure that the values entered are in consistent units. Click ***OK***.

We see in the *Real Constants* menu that the constant set that we just created is "Set 1". So, when we mesh the geometry later on, we'll use the reference no. 1 to assign this constant set.



Click ***Close***.

Go to Step 3: Specify material properties

See and rate the complete Learning Module

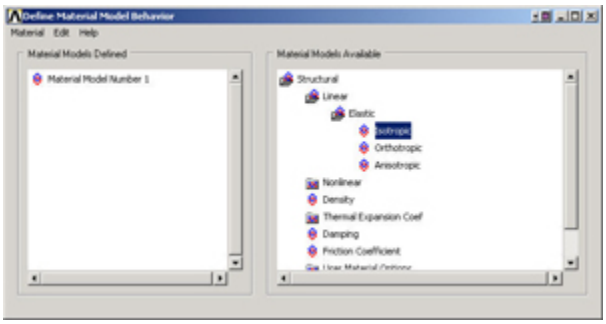
Go to all ANSYS Learning Modules

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results
Problem Set 1
Problem Set 2

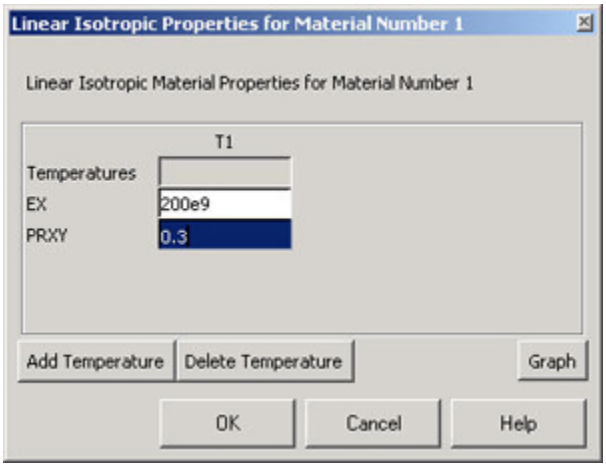
Step 3: Specify material properties

Main Menu > Preprocessor > Material Props > Material Models

In the ***Material Models Available*** Frame of the *Define Material Model Behavior* window, double-click on ***Structural***, ***Linear***, ***Elastic***, and ***Isotropic***.



Enter `200e9` for *Young's Modulus* ***EX***.
Enter `0.3` for *Poisson's Ratio* ***PRXY***.



Click ***OK***. This completes the specification of *Material Model Number 1*. When we mesh the geometry later on, we'll use the reference no. 1 to assign this material model. Close the *Define Material Model Behavior* menu.

Save your work

Utility Menu > File > Save as Jobname. db

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 this "database file" in your working directory. You can restart from your last save at any time using

Utility Menu > File > Resume Jobname. db or ANSYS Toolbar > RESUM_DB

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.

Go to Step 4: Specify geometry

See and rate the complete Learning Module

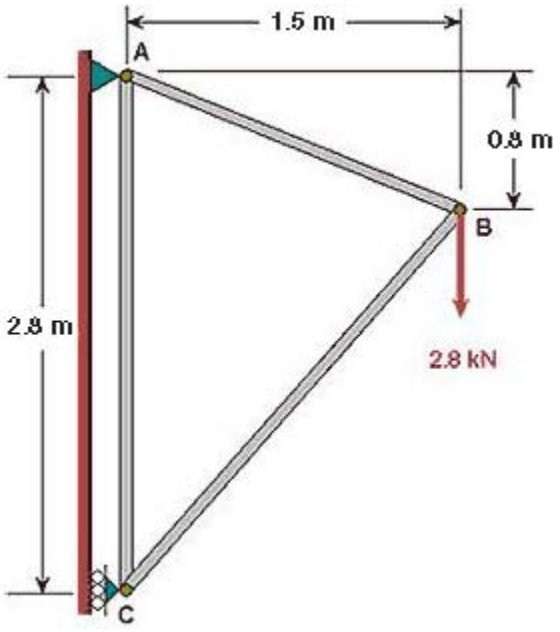
Go to all ANSYS Learning Modules

Problem Specification
Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results
Problem Set 1
Problem Set 2

Step 4: Specify geometry

Overview

Since we are using the 2D Spar element, we can represent each truss member by a line. A line can be created by joining two keypoints (ANSYS terminology for vertices). So we'll need three keypoints, located at A, B and C in the figure below. We'll locate the origin of the coordinate system at C and number the keypoints at A, B and C as 1, 2 and 3, respectively.



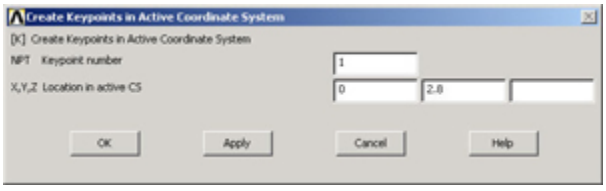
Create Keypoints

Main Menu > Preprocessor > Modeling > Create > Keypoints > 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 Keypoints in Active Coordinate System* dialog box,

Enter `1` for ***Keypoint number***
Enter `0` for X and `2.8` for Y
(The Z value defaults to zero)
Click ***Apply*** (which accepts the input and then brings back the dialog box for further input)



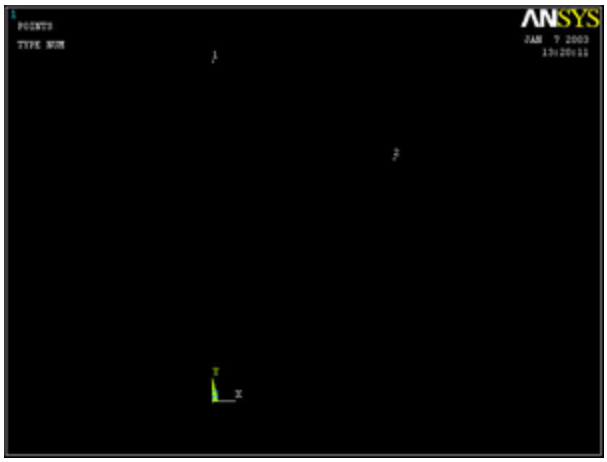
Note that you can move to the next field using the *Tab* key.

Enter 2 for ***Keypoint number***
Enter `1.5` for X and `2.0` for Y
Click ***Apply***

Enter 3 for ***Keypoint 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 keypoints will now be displayed in the Graphics window along with a triad that indicates the origin of the coordinate system (coincident with keypoint 3 in our case) and the axes.



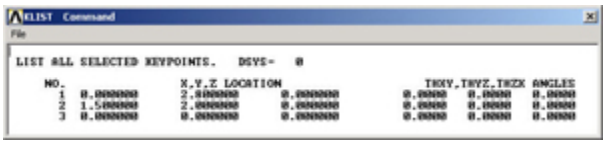
Higher Resolution Image

Check Keypoints

To check if the keypoints have been created correctly:

Utility Menu > List > Keypoints > Coordinates only

This brings up a window listing the coordinates and rotation angles for the keypoints. Verify that you have the following:



You can rotate the coordinate system associated with each keypoint and that is what the rotation angles THXY, THYZ and THZX refer to. In our case, we don't need to rotate the keypoint coordinate system and so the rotation angles are identically zero.

Close the window listing the keypoints.

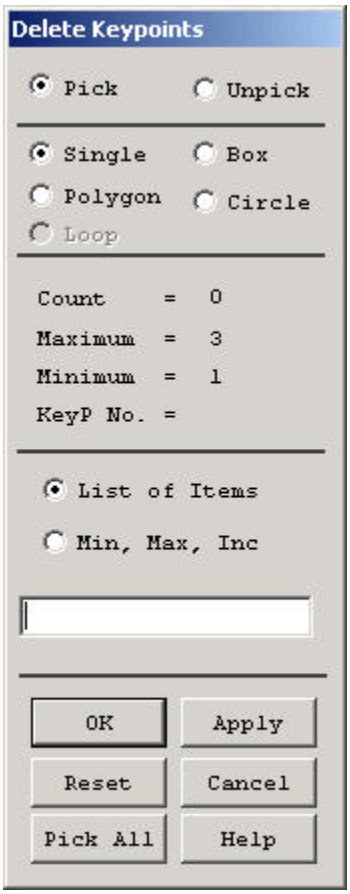
Close the *Keypoints* and *Create* menus.

Modifying Keypoints (If Necessary)

If you are like me, you made a mistake while creating keypoints and cursed that there is no undo button. To correct your mistake(s), you can delete keypoints and re-create them. In case you need to delete a keypoint, do the following:

Main Menu > Preprocessor > Modeling >Delete > Keypoints

This brings up the so-called pick menu.



Click on the keypoint you want to delete. A square appears around that keypoint indicating that it is selected. (Repeat for other keypoints as necessary.) Click ***OK*** in the pick menu.

You should see the keypoint disappear in the Graphics window. You can also check that the keypoint has been deleted using **Utility Menu > List > Keypoints**. You can then re-create the keypoint.

Save your work

Once you have successfully created the keypoints, save your work using

Toolbar > SAVE_DB

This is a short-cut for

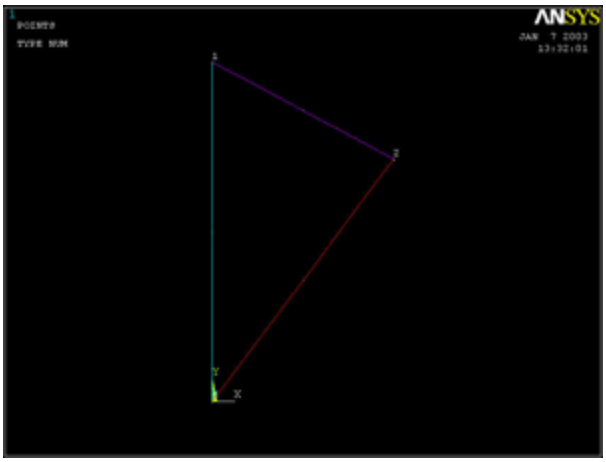
Utility Menu > File > Save as Jobname. db

Create Lines from Keypoints

Main Menu > Preprocessor > Modeling > Create > Lines > Lines >In Active Coord

This will bring up the *Lines in Active Coord* pick menu.

To create the line between keypoints 1 and 3, click on keypoint 1 and then keypoint 3. Similarly, create lines between keypoints 1 & 2 and keypoints 2 & 3.



Higher Resolution Image

Close the *Lines in Active Coord* pick menu by clicking on ***Cancel***.

Close the *Lines and Create* menus.

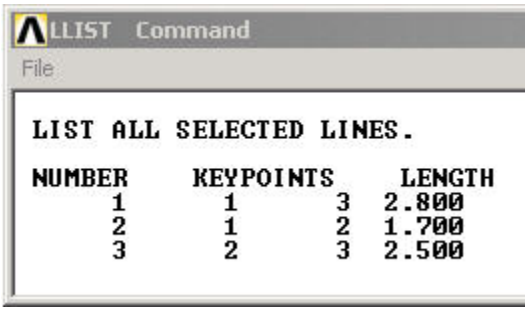
Check Lines

Take a look at the list of lines that have been created:

Utility Menu > List > Lines

Click ***OK*** to accept the default output format.

The first four columns in the list should be:



The order of the keypoints for a line doesn't matter i.e. line *l1* could go from keypoint *k1* to *k2* or equivalently, *k2* to *k1*.

Close the window listing lines.

Modifying Lines (If Necessary)

If a line doesn't look right, you can delete and re-create it. To delete a line:

Main Menu > Preprocessor > Modeling > Delete > Lines Only

This brings up the pick menu.

Click on the line you want to delete in the Graphics window. Click ***OK***. This deletes the line.

Save your work

Once you have successfully created the lines, click on **Toolbar > SAVE_DB** to save the database.

Go to Step 5: Mesh geometry

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results
Problem Set 1
Problem Set 2

Step 5: Mesh geometry

Each truss member can be represented as a 2D Spar element. We'll use the *MeshTool* to mesh the geometry with this element.

To bring up the *MeshTool*, select

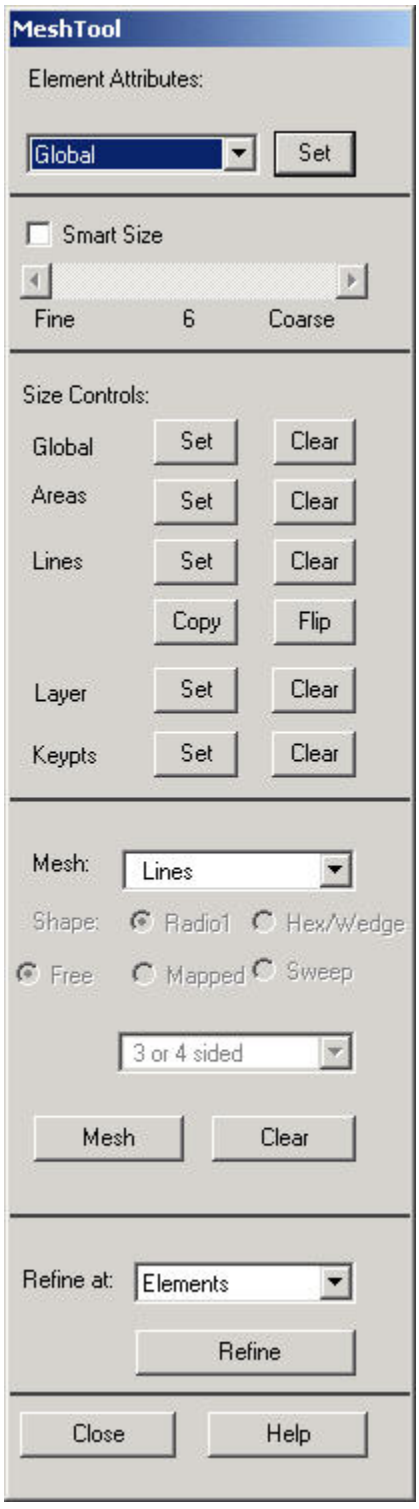
Main Menu > Preprocessor > Meshing > MeshTool

The *MeshTool* is used to control and generate the mesh.

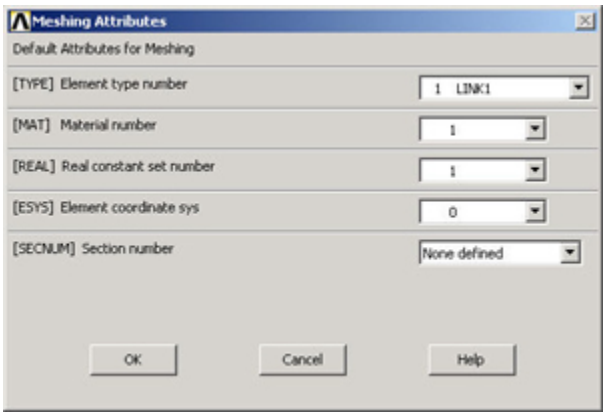
Set Meshing Parameters

We'll now specify the element type, real constant set and material property set to be used in the meshing. Since we have only one of each, we can assign them to the entire geometry using the ***Global*** option under ***Element Attributes***.

Make sure ***Global*** is selected under ***Element Attributes*** and click on ***Set***.



This brings up the *Meshing Attributes* menu. You will see that the correct element type, material number and real constant set are already selected since we have only one of each.



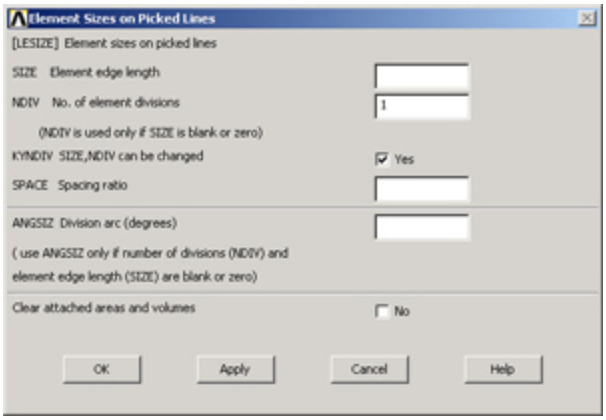
Click ***OK***. ANSYS now knows what element type (and associated constants) and material type to use for the mesh.

Set Mesh Size

Since a *LINK1* element is equivalent to a truss member, we will specify that we want only one element per line. This is a subtle point and also very unusual; in most problems, you want to subdivide your part into many elements.

In the *MeshTool*, under ***Size Controls*** and ***Lines*** , click ***Set***.

In the pick menu that comes up, click ***Pick All*** (since we want the specification of mesh size to apply to all lines in the geometry). This brings up the *Elemen t Sizes on Picked Lines menu*. Specify ***No. of element divisions*** to be 1. Click ***OK***. ANSYS will now use 1 element to mesh each line.



Mesh Lines

In the *MeshTool*, make sure ***Lines*** is selected in the drop-down list next to ***Mesh***. This means the geometry components to be meshed are lines (as opposed to areas or volumes, as we'll see later). Click on the ***Mesh*** button.

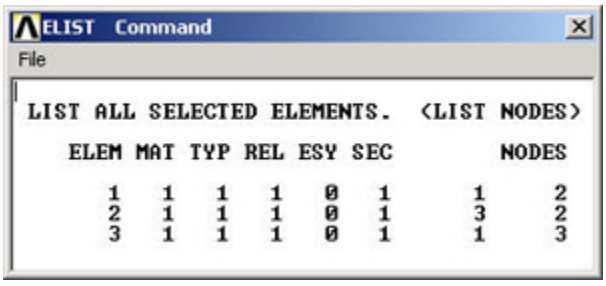
This brings up the pick menu. Since we want to mesh all lines, click on ***Pick All***. The lines have been meshed. This is reported in the *Output Window* (usually hiding behind the *Graphics Window*):

NUMBER OF LINES MESHED = 3
MAXIMUM NODE NUMBER = 3
MAXIMUM ELEMENT NUMBER = 3

Close the *MeshTool*.

View Element List

Utility Menu > List > Elements > Nodes + Attributes



This table says that *Element 1* is of *material type 1* and *element type 1* and is attached to nodes 1 and 2 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. Also, the order of element numbering is not important since it is for internal bookkeeping.

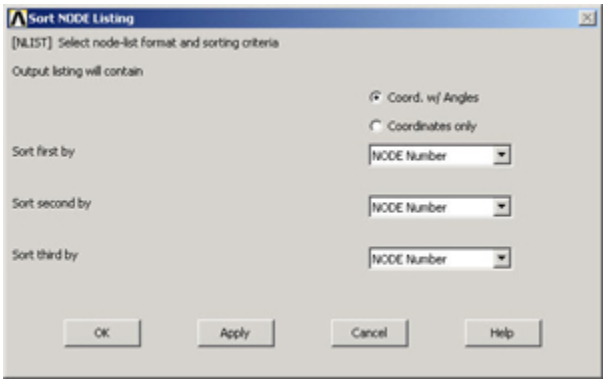
Close the window listing the elements.

View Node Location

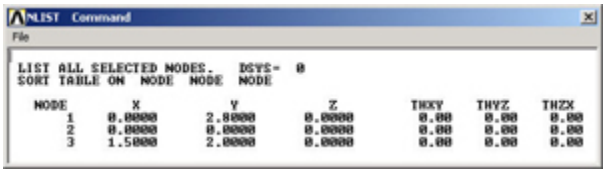
In order to see where the nodes are located, you can look at the list of nodes.

Utility Menu > List > Nodes

In *Sort NODE Listing* menu, click ***OK*** to accept defaults.



My list of nodes looks like this:



From the node and element lists, one can conclude that in this case:

Node 1 is pin A
Node 2 is pin C
Node 3 is pin B

Element 1 is member AC
Element 2 is member AB
Element 3 is member BC

Your own node and element numbering might be different from this and you would have to account for this while interpreting results in the postprocessing step.

Close the window listing the nodes.

Save your work

Toolbar > SAVE_DB

Go to Step 6: Specify boundary conditions

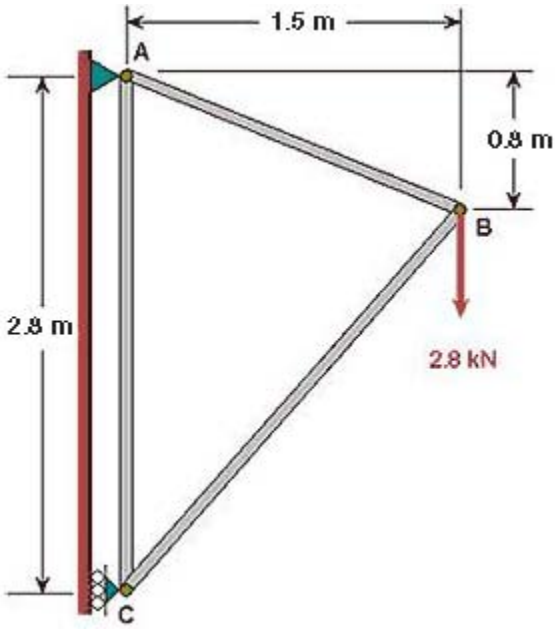
See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results
Problem Set 1
Problem Set 2

Step 6: Specify boundary conditions

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". We can apply the loads either to the geometry model or to the finite-element model (that is to the elements and nodes directly). The advantage of the former is that one doesn't have to re-specify the constraints on changing the mesh. So we'll apply the constraints to the geometry i.e. to the keypoints.

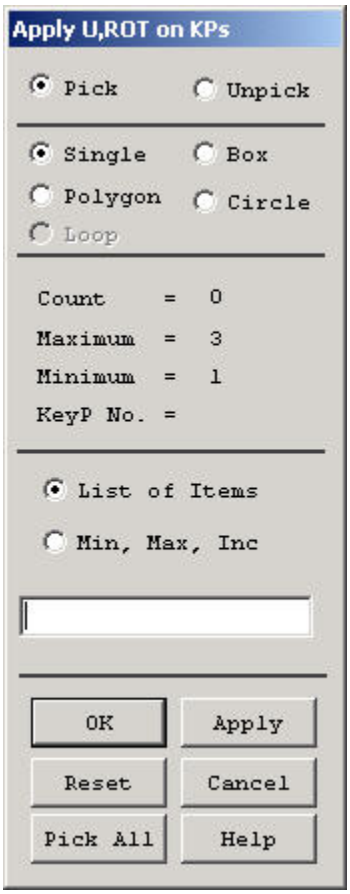


You can see from the diagram that the pin at A is constrained in x and y directions; or equivalently, keypoint 1 is constrained such that its *UX* and *UY* displacements are zero. Similarly, keypoint 3 is constrained such that its *UX* displacement is zero.

Apply Displacement Constraints

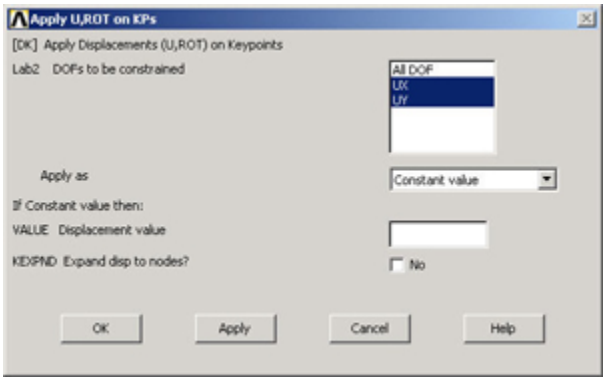
Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Keypoints

This brings up the *Apply U, ROT on KPs* pick menu.



In the *Graphics* window, click on *keypoint 1*;
This will draw a small square around keypoint 1 to indicate that it's been picked.

In the pick menu, click ***Apply***. The following menu shows up.

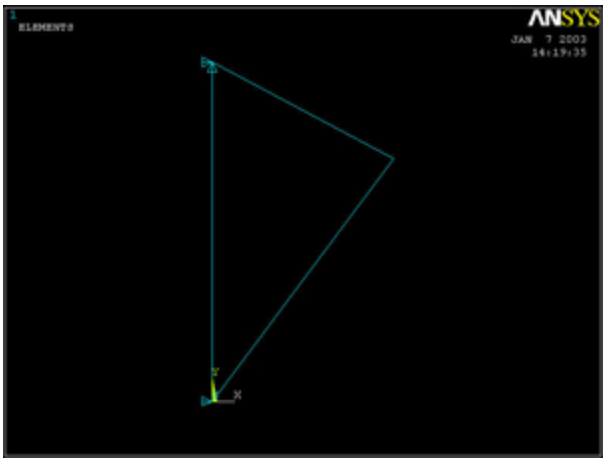


Since we want to constrain ***UX*** as well as ***UY*** to zero at keypoint 1, select both ***UX*** and ***UY*** from items in ***DOFs to be constrained*** list. Since the ***Displacement value*** is zero by default, leave that field empty. Click on ***Apply***.

You will see triangle symbols appear in the Graphics window indicating that both *UX* and *UY* DOFs are constrained at keypoint 1.

Next we apply the displacement constraint at keypoint 3. In the *Graphics* window, click on keypoint 3. In the pick menu, click ***Apply***. Select only ***UX*** from items in ***DOFs to be constrained*** list. Click ***OK***.

You will see a triangle symbol appear indicating that only the *UX* DOF is constrained at keypoint 3.



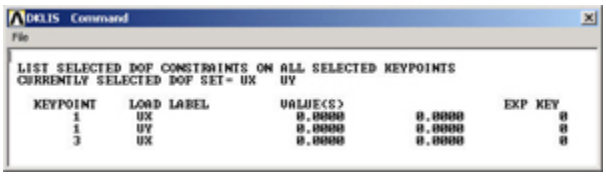
Close the *Displacement* and *Apply* menus.

List Displacement Constraints

You can verify the displacement constraints on the model by listing them.

Utility Menu > List > Loads > DOF constraints > On All Keypoints

This brings up a window with the constraint information.



If you made a mistake in applying a constraint, you can delete and reapply it. You can delete a constraint using

Main Menu > Preprocessor > Loads > Define Loads >Delete > Structural > Displacement > On Keypoints

Alternately, you can resume from your last save and continue from there.

Close the window listing the constraint information.

Save the database: **Toolbar > SAVE_DB**

Apply Loading

We have only one load to apply on the structure, namely, the 2800 N force in the negative y-direction at keypoint 2 (see figure 1).

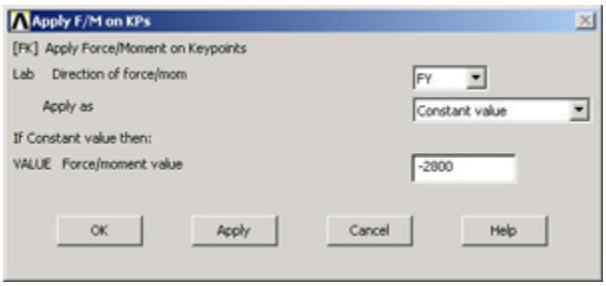
Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural >Force/Moment > On Keypoints

This brings up the *Apply F/M on KPs* pick menu.

In the *Graphics* window, click on keypoint 2; then in the pick menu, click ***OK***.

In the menu 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.

Close the F*orce/ Moment, Apply, Loads* and *Preprocessor* menus.

Save the database: **Toolbar > SAVE_DB**

Go to Step 7: Solve!

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results
Problem Set 1
Problem Set 2

Step 7: Solve!

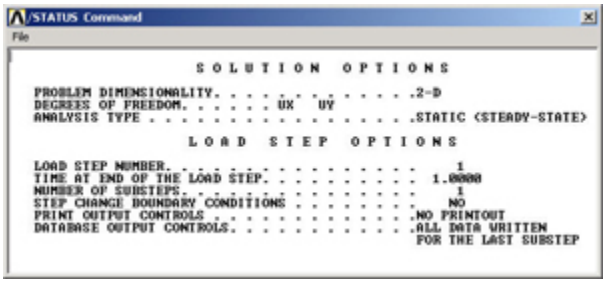
Enter Solution Module

Main Menu > Solution > Solve > 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 Problem

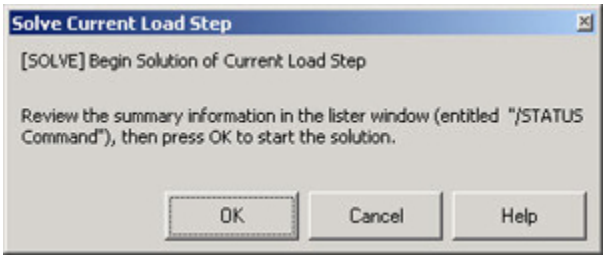
Review the information in the */STAT Command window*. This is a summary of the problem that ANSYS is about to solve.



Close this window.

Perform Solution

Click ***OK*** in *Solve Current Load Step* menu.



ANSYS performs the solution and a window should pop up saying "Solution is done!". Congratulations! You just obtained your first ANSYS solution.

Close the window.

In preparation for the postprocessing step to be undertaken next, exit the solution module by closing the *Solution* menu.

Verify that ANSYS has created a file called *truss.rst* in your working directory. This file contains the results of the (previous) *solve*. The *rst* extension in the filename stands for results from a structural analysis. The *truss.db* file contains only steps 1-6. To resume your work subsequent to exiting ANSYS, you'll have to first resume from the *jobname.db* file and then read in the results from the *jobname.rst* file using

Main Menu > General Postproc > Read Results > First Set

This is one of the many ANSYS quirks you'll encounter as you work with the program.

Go to Step 8: Postprocess the results

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results
Problem Set 1
Problem Set 2

Step 8: Postprocess the results

Postprocessing is the step where we look at and analyze the results obtained from the ANSYS solution.

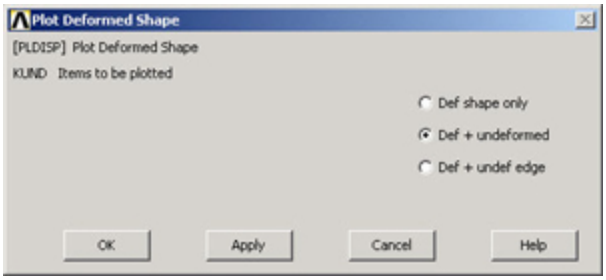
Enter the *General Postprocessing* module:

Main Menu > General Postproc

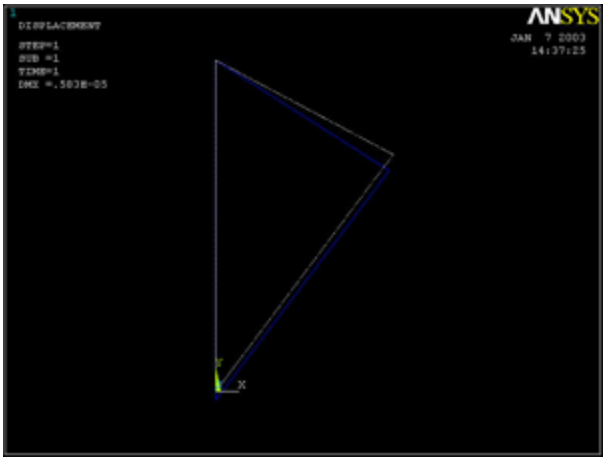
Plot Deformed Shape

Main Menu > General Postproc > Plot Results > Deformed Shape

Select ***Def + undeformed*** and click ***OK***.



This plots the deformed and undeformed shapes in the *Graphics* window.



The deformed shape is shown as a solid line and the undeformed shape as a dotted line. The maximum displacement *DMX* is 0.583E-05m as reported in the *Graphics* window. This is small but plausible. Note that the deformation is magnified in the plot so as to be easily visible.

To save the deformation plot in a file, use **Utility Menu > PlotCtrls > Hard Copy > To File**. Select the file format you want and type in a filename of your choice under ***Save to:*** and click ***OK***. The file will be created in your working directory. You can print out this file as necessary.

Animate the deformation:

Utility Menu > PlotCtrls > Animate > Deformed Shape

Select ***Def + undeformed*** and click ***OK***. Select ***Forward Only*** in the *Animation Controller*.

Node 1 (Pin A) doesn't move and node 2 (Pin C) moves only in the vertical direction. Node 3 (Pin B) moves more or less in the direction of the applied force. The deformation of the structure agrees with the applied boundary conditions and matches with what one would expect from intuition. Stop the animation and close the animation control window.

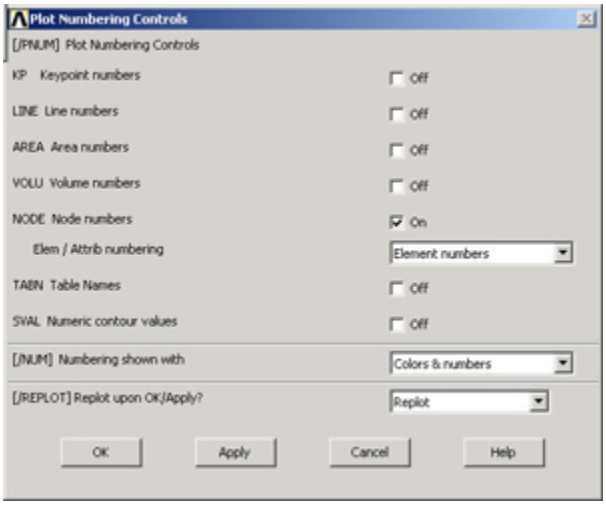
Turn On Node and Element Numbers

In order to interpret the results that ANSYS reports, it's useful to turn on the node and element numbers in the *Graphics* window.

Utility Menu > PlotCtrls > Numbering

The *Plot Numbering Controls* menu is used to control the numbering of the various entities in a finite-element model.

Turn on ***Node numbers***. Under ***Elem/Attrib numbering***, select ***Element numbers***. Click ***OK***.



The node and element numbers will now appear in the *Graphics* window.

List Forces in Truss Members

Bring up the help page for LINK1 element:

Utility Menu > Help > Help Topics

Under the ***Contents*** tab, select

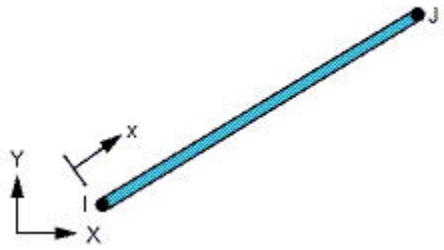
Release 11.0 Documentation for ANSYS > Elements Reference > Element Library > LINK1

In the *LINK1* help page, scroll down to *Table 1.1: LINK1 Element Output Definitions*. This table lists all the output items you can get for the LINK1 element. Let's scour this table for the element output that corresponds to the force in a truss member. We see the following item in the table that looks like a promising candidate:

MFORX: Member force in the element coordinate system X direction

The figure at the top of the *LINK1* help page, reproduced below, shows that the x-direction in the *element coordinate system* is along the element.

Figure 1.1 LINK1 Geometry



Thus, *MFORX* is basically the axial force in the element.

So how do we get the *MFORX* values for our three elements from ANSYS? ANSYS has a quirky way of doing this as we shall see. If you scroll down the help page further, you'll see *Table 1.2 LINK1 Item and Sequence Numbers* with the following entry for *MFORX*:

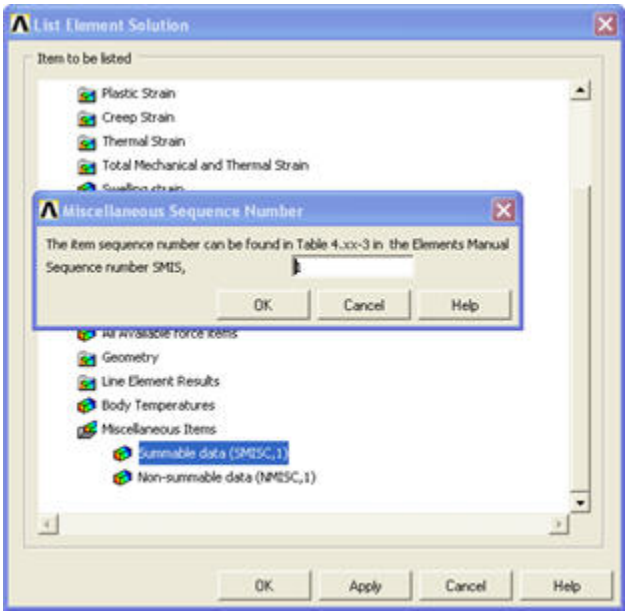
MFORX SMISC 1

The output data for any element type is broken down into item groups with *SMISC* being one of the groups (*SMISC* stands for "Summable Miscellaneous" items). Each item within a group such as *SMISC* has an identifying "sequence" number. So, as per the above entry in Table 1.2, *MFORX* is an item in the *S MISC* group with a sequence number 1. In effect, it's entry #1 in the SMISC group for the LINK1 element.

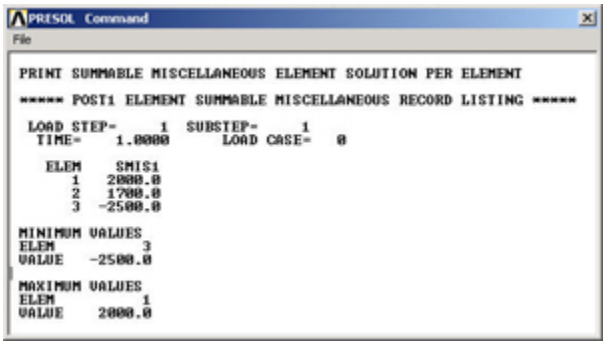
Minimize the help window. To list *MFORX* values, select:

Main Menu > General Postproc > List Results > Element Solution

Under ***Element Solution***, select ***Miscellaneous Items > Summable data (SMISC,1)***. Since *MFORX* is sequence number 1 in the *SMISC* group, enter 1 next to ***Sequence number SMIS*** in the editable field. Click OK. Click OK in the *List Element Solution* window.



This brings up a window with the axial forces in the elements. Positive values indicate tension and negative values compression.



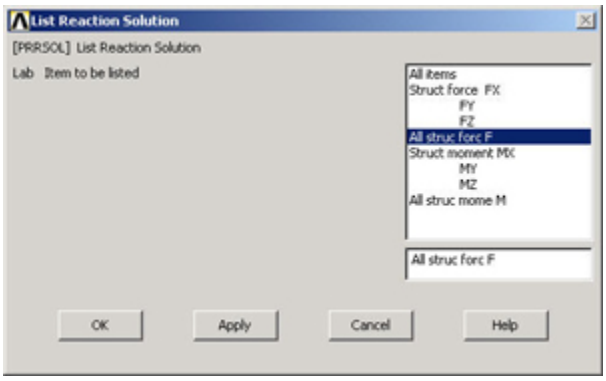
You can also *plot* the items listed under *Element Output Definitions* using the sequence number.

In most cases, you plot stresses using **Main menu > General Postproc > Plot Results > Contour Plot >Nodal Solu**. But for line elements like *LINK1*, this doesn't work and you'll get zero values for the stresses. So you'll have to use the sequence numbers to make stress plots for line elements.

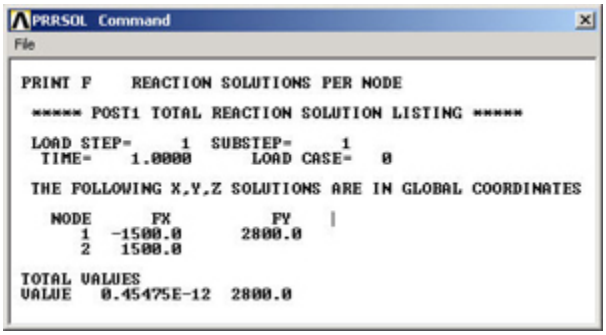
List Reaction Forces at Nodes

Main Menu > General Postproc > List Results > 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.



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

Close the *PRRSOL Command* window.

Go to Step 9: Validate the results

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results
Problem Set 1
Problem Set 2

Step 9: Validate the results

Do not assume that if you are able to obtain a solution from ANSYS, it is bound to be correct. It is ***very important*** that you take the time to check the validity of your solution. This section leads you through some of the steps you can take to validate your solution.

Simple Checks

- Does the deformed shape look reasonable and agree with the applied boundary conditions? We checked this in step 8.
- Do the reactions at the supports balance the applied forces for static equilibrium? We checked this also in step 8.

Refine Mesh

The results obtained from FEA analysis depend on the mesh. An important step in the analysis is to make sure that the mesh resolution is adequate for the desired level of accuracy. This is done by refining the mesh and comparing results obtained with different levels of mesh resolution.

In our truss example, however, a truss member has to be modeled as a ***single*** *LINK1* element. If we use multiple *LINK1* elements to model a single truss member, these elements can rotate freely with respect to each other since they are essentially linked through pin joints. This violates physical reality and is one of the few cases where we'll avoid refining the mesh since it leads to an incorrect result.

Compare with Theory

Results should be compared with appropriate theoretical results whenever possible. In most cases, one would use theory to obtain order-of-magnitude estimates rather than to make a head-to-head comparison since presumably FEA is being used because a theoretical solution is not available. In this case, however, one can easily determine the forces in the truss members using the method of joints from statics. I'd recommend that you take a few minutes during commercials on your favorite TV show to calculate the forces and compare them with your ANSYS results.

Exit ANSYS

Utility Menu > File > Exit

Select ***Save Everything*** and click ***OK***.

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 using **Utility Menu > Help**. There are tutorials available in the documentation which are also useful.

Go to Problem Set 1

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results
Problem Set 1
Problem Set 2

Problem Set 1

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

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 will change?

Re-solve the truss problem with the modified constraints. Fom your ANSYS solution:
1. List 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 > Save As

2. Determine the force in each truss member and whether the member is in tension or compression.

Tips: Start from your ANSYS solution to the truss tutorial problem but use a different job name as follows:

- Start the ANSYS Product Launcher. Specify the same directory as for the tutorial but use a *different jobname*. Once ANSYS comes up, select: **Utility Menu > File > Resume from**. Choose *truss.db* and click **OK**.

Note that you can delete constraints using:

Main Menu > Preprocessor > Loads > Define Loads > Delete

It works similar to how you apply loads.

You can plot the displacement constraints in the Graphics window as follows:

Utility Menu > Pltctrls > Symbols

Select ***All Applied BCs*** for ***Boundary condition symbol***. Click ***OK***.

You might have to use :

Utility Menu > Plot > Replot

or

Utility Menu > Plot > Multi-plots

for the constraint symbols to appear in your plot.
Go to Problem Set 2

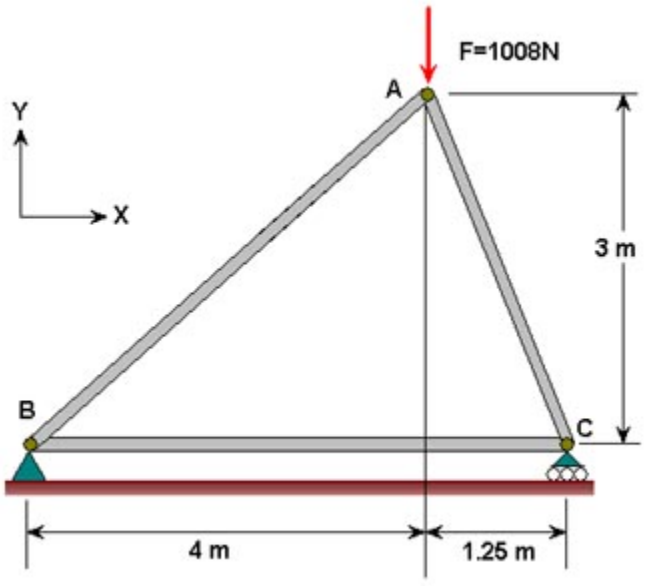
See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results
Problem Set 1
Problem Set 2

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, Young's modulus is $200x10^9$ N/m^2 and Poisson's ratio is 0.3. Verify your results by calculating the forces manually.



Results

Determine the following:

1. Listing of the reactions from the ANSYS solution.

2. Listing of the element forces from the ANSYS solution. Calculate and determine the forces in each member and whether the member is in tension or compression from this ANSYS result.

3. Using your pencil-and-paper calculations verifying the ANSYS results for the member and reaction forces.

See and rate the complete Learning Module

Go to all ANSYS Learning Modules