

Cantilever Beam - Pre-Analysis & Start-Up (OLD)

Author: John Singleton, Cornell University

Problem Specification

1. Pre-Analysis & Start-Up

2. Geometry

3. Mesh

4. Physics Setup

5. Numerical Solution

6. Numerical Results

7. Verification & Validation

Exercises

Comments

Pre-Analysis & Start-Up

Pre-Analysis

We'll start by carrying out hand calculations to predict the expected maximum bending stress and deformation. We'll later compare the ANSYS values to these hand-calculation results.

Calculate the Maximum Bending Stress at x=0m

The maximum bending stress is calculated using Euler-Bernoulli beam theory.

$$\sigma_{max} = \frac{M(x)Y}{I}$$
$$I = \frac{bh^3}{12}$$
$$\sigma_{max} = \frac{(8000N)(4m) \left(\frac{.346m}{2} \right)}{\frac{(0.346m)(0.346m)^3}{12}} = 4.635MPa$$

Calculate the Total Deformation at x=4m

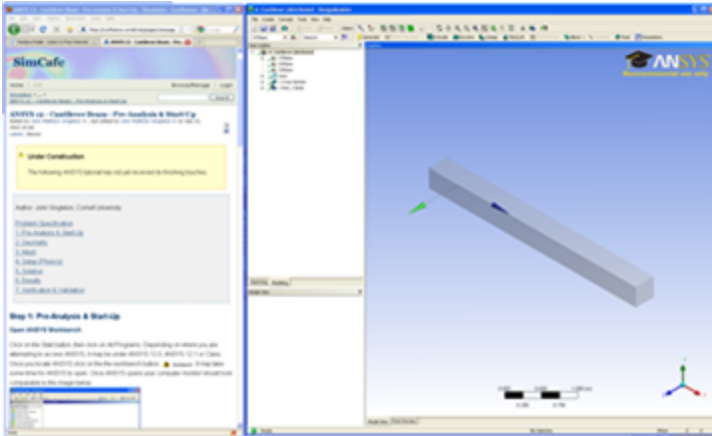
$$u'' = \frac{M}{EI}, \quad u'(0) = 0, \quad u(0) = 0$$
$$M = PL \left(1 - \frac{x}{L} \right)$$
$$u'' = \frac{PL}{EI} \left(1 - \frac{x}{L} \right)$$
$$u' = \int u'' dx = \int \frac{PL}{EI} \left(1 - \frac{x}{L} \right) dx = \frac{PL}{EI} \left(x - \frac{x^2}{2L} \right) + C_1$$
$$u'(0) = 0 \quad \therefore \quad u' = \frac{PL}{EI} \left(x - \frac{x^2}{2L} \right)$$
$$u = \int u' dx = \int \frac{PL}{EI} \left(x - \frac{x^2}{2L} \right) dx = \frac{PL}{EI} \left(\frac{x^2}{2} - \frac{x^3}{6L} \right) + C_2$$
$$u(0) = 0 \quad \therefore \quad u = \frac{PL}{EI} \left(\frac{x^2}{2} - \frac{x^3}{6L} \right)$$
$$u(4m) = \frac{(8000N)(4m)}{(2.8 \times 10^{10}) \left(\frac{(0.346m)(0.346m)^3}{12} \right)} \left(\frac{(4m)^2}{2} - \frac{(4m)^3}{6(4m)} \right) = 0.005103m$$

[Click Here For the Deformation Calculation in Higher Resolution](#)

Start-Up

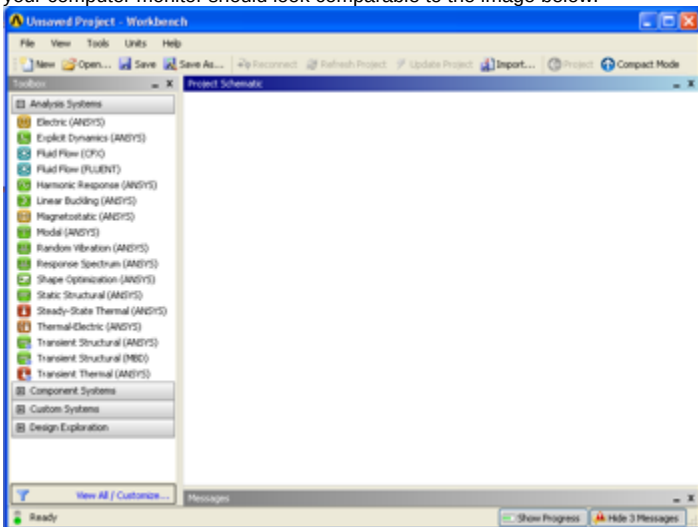
Optimizing Monitor Real Estate

This tutorial is specially configured, so the user can have both the tutorial and ANSYS open at the same time as shown below. It will be beneficial to have both ANSYS and your internet browser displayed on your monitor. Your internet browser should consume approximately one third of the screen width while ANSYS should take the other two thirds.



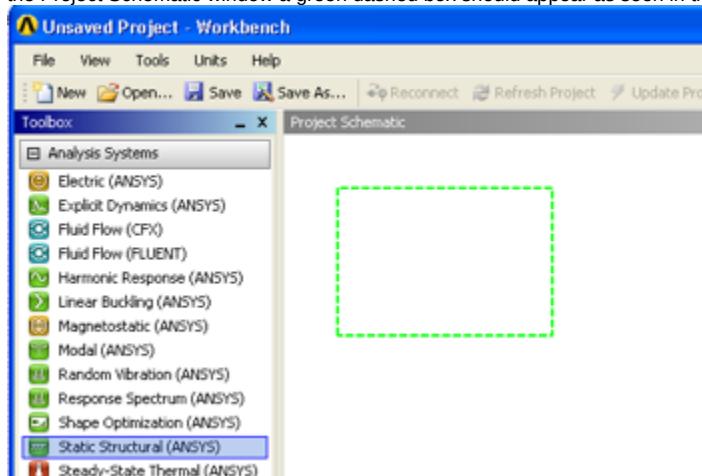
Open ANSYS Workbench

Click on the Start button, then click on All Programs. Depending on where you are attempting to access ANSYS, it may be under ANSYS 13.0, ANSYS, or Class. Once you locate ANSYS click on the the workbench button,  Workbench. It may take some time for ANSYS to open. Once ANSYS opens your computer monitor should look comparable to the image below.

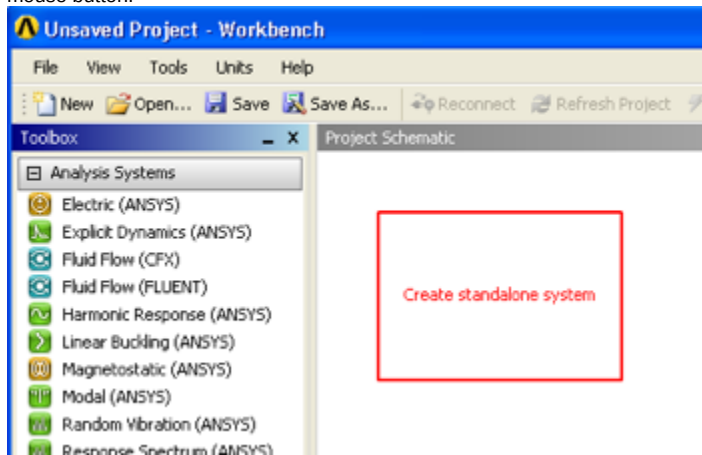


Static Structural Analysis System

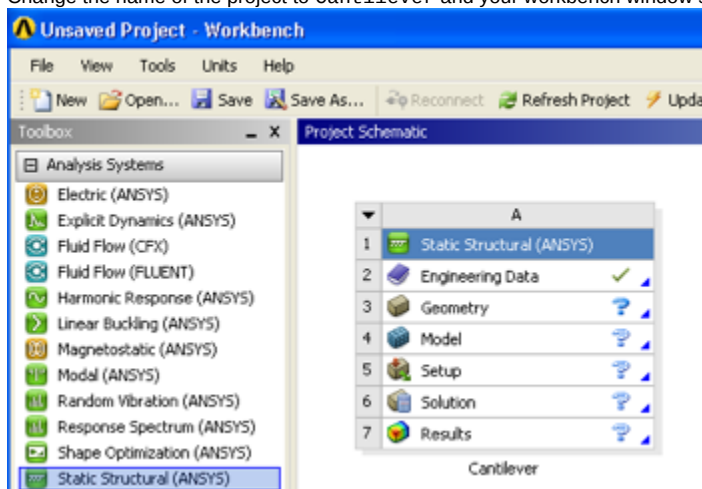
The problem at hand is a static structural problem, so click and hold down the mouse button on the Static Structural (ANSYS) button,  Static Structural (ANSYS), and drag it over to the project schematic window. When you begin to drag the Static Structural (ANSYS) button over to the Project Schematic window a green dashed box should appear as seen in the image below.



Drag the Static Structural (ANSYS) button into the green box until it turns red and has the text "Create standalone system" within it, then release the mouse button.

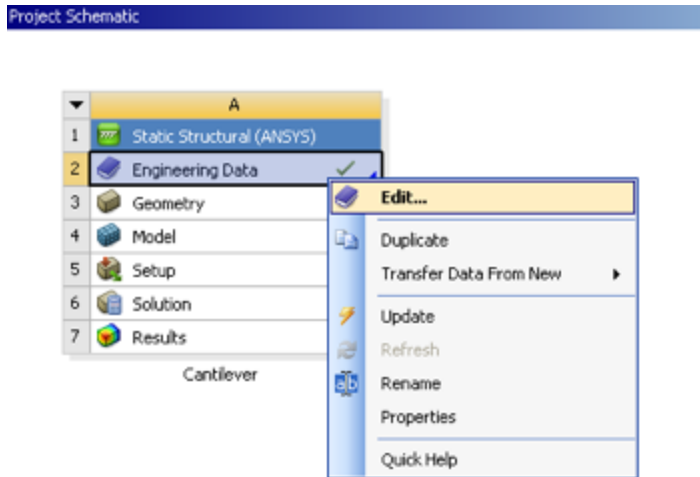


Change the name of the project to Cantilever and your workbench window should look similar to the image below.

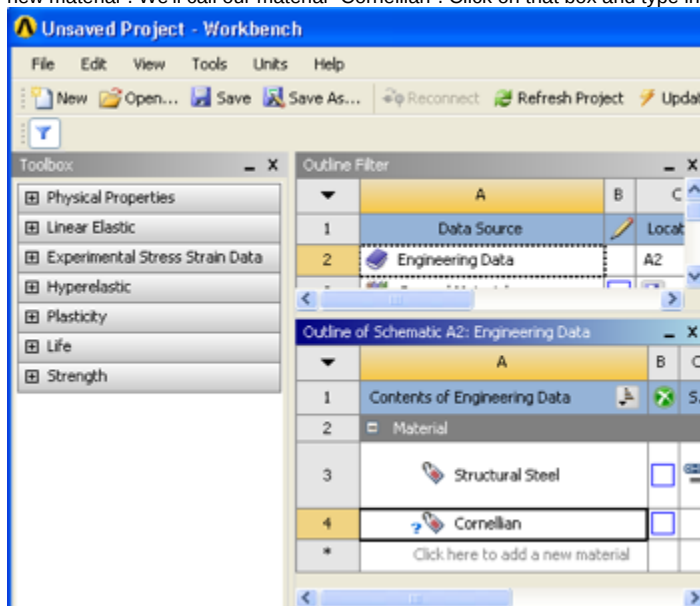


Engineering Data

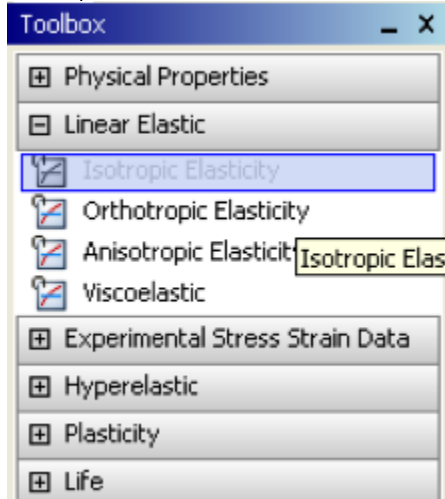
The specific properties of our material needs to be inputted into ANSYS. Start by right clicking on *Engineering Data* and then clicking on *Edit..* as seen below.



At this point a new window will open. Under *Outline of Schematic A2: Engineering Data* there will be a box with text inside that says "Click here to add a new material". We'll call our material "Cornellian". Click on that box and type in Cornellian and then press enter.



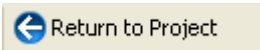
Now, expand the *Linear Elastic* tab on the left and double click on *Isotropic Elasticity*.



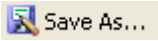
Now, the properties of "Cornellian" need to be entered. Set the Young's Modulus to 2.8×10^{10} Pascals and set Poisson's Ratio to 0.4. Note that the stiffness matrix for the beam element is independent of the Poisson's Ratio. Hence the solution will not change if a different Poisson's Ratio is used.

Properties of Outline Row 4: Cornellium					
	A	B	C	D	E
1	Property	Value	Unit		
2	Isotropic Elasticity				
3	Derive from	Young's...			
4	Young's Modulus	2.8E+10	Pa		
5	Poisson's Ratio	0.4			
6	Bulk Modulus	4.6667E+10	Pa		
7	Shear Modulus	1E+10	Pa		

The material "Cornellian" will be assigned to our model in a later step. Here we have just input its Young's Modulus and Poisson's ratio.

At this point the project can be returned to. Click on the Return to Project button,  at the top.

Saving

It would be of best interest, to save the project at this point. Click on the "Save As..." button, , which is located on the top of the Workbench window. Save the project as "Cantilever" in a suitable folder. When you save in ANSYS a file and a folder will be created. For instance if you save as "Cantilever", a "cantilever.wbpj" file and a folder called "cantilever_files" will appear. In order to reopen the ANSYS files in the future you will need both the ".wbpj" file and the folder. If you do not have BOTH, you will not be able to access your project.

[Go to Step 2: Geometry](#)

[Go to all ANSYS Learning Modules](#)