

# ANSYS 12 - Beam (2D Element) - Step 1

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

[Problem Specification](#)

- [1. Pre-Analysis & Start-Up](#)
- [2. Geometry](#)
- [3. Mesh](#)
- [4. Setup \(Physics\)](#)
- [5. Solution](#)
- [6. Results](#)
- [7. Verification & Validation](#)

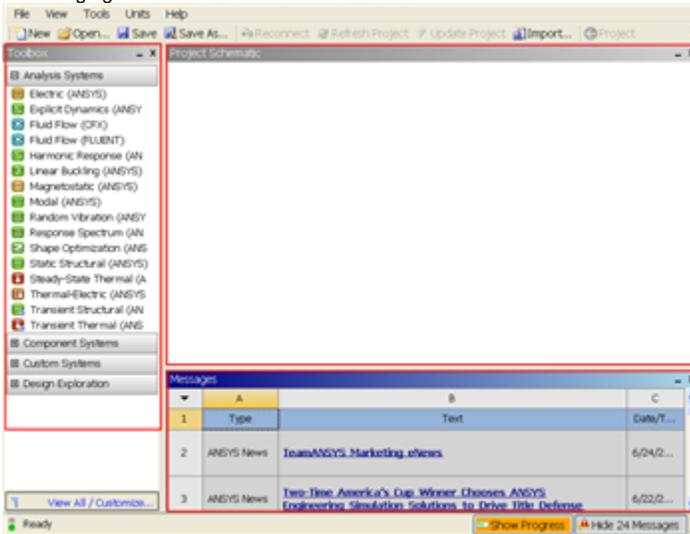
## Step 1: Pre-Analysis & Start-Up

### Start ANSYS Workbench

We start our simulation by first starting the ANSYS workbench.

**Start > All Programs > ANSYS 12.1 > Workbench**

Following figure shows the workbench window.



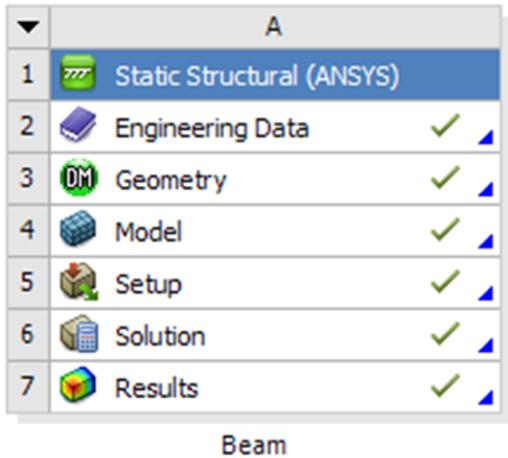
At the left hand side of the workbench window, you will see a toolbox full of various analysis systems. In the middle, you see an empty work space. This is the place where you will organize your project. At the bottom of the window, you see messages from ANSYS.

### Select Analysis Systems

[Select Analysis System Demo](#)

Since our problem involves static analysis, we will select the **Static Structural (ANSYS)** component on the left panel. Left click (and hold) on **Static Structural (ANSYS)**, and drag the icon to the empty space in the **Project Schematic**.

Since we selected Static Structural (ANSYS), each cell of the system corresponds to a step in the process of performing the ANSYS Structural analysis. Right click on **Static Structural ANSYS** and **Rename** the project to **Beam**.



Now, we just need to work out each step from top down to get to the results for our solution.

- We start by preparing our geometry
- We use geometry to generate a mesh
- We setup the physics of the problem
- We run the problem in the solver to generate a solution
- Finally, we post process the solution to gain insight into the results

### Specify Material Properties

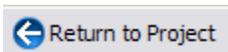
We will first specify the material properties of the crank. The material has an Young's modulus  $E=2.8 \times 10^7$  psi and Poisson's ratio  $\nu=0.3$ .

In the Crank cell, double click on **Engineering Data**. This will bring you to a new page. The default material given is **Structural Steel**. We will use this material and change the Young's modulus and Poisson's ratio.

Left click on **Structural Steel** once and you will see the details of Structural Steel material properties under **Properties of Outline Row 3: Structural Steel**. Expand **Isotropic Elasticity**, change **Young's Modulus** and **Poisson's Ratio** to  $E=7.9 \times 10^{10}$  pa and  $\nu=0.3$ . Remember to check that you use the correct unit.

| Properties of Outline Row 3: Structural Steel |                                  |         |                    |     |
|---|----------------------------------|---------|--------------------|-----|
|   | A                                | B       | C                  | D E |
| 1   | Property                         | Value   | Unit               |     |
| 2   | Density                          | 7850    | kg m <sup>-3</sup> |     |
| 3   | Coefficient of Thermal Expansion |         |                    |     |
| 6   | Isotropic Elasticity             |         |                    |     |
| 7   | Young's Modulus                  | 7.3E+10 | Pa                 |     |
| 8   | Poisson's Ratio                  | 0.3     |                    |     |
| 9   | Alternating Stress Mean Stress   | Tabular |                    |     |
| 13  | Strain-Life Parameters           |         |                    |     |
| 21  | Tensile Yield Strength           | 2.5E+08 | Pa                 |     |
| 22  | Compressive Yield Strength       | 2.5E+08 | Pa                 |     |
| 23  | Tensile Ultimate Strength        | 4.6E+08 | Pa                 |     |
| 24  | Compressive Ultimate Strength    | 0       | Pa                 |     |

[Higher Resolution Window](#)



Press the **Return to Project** to return to Workbench **Project Schematic** window.

[Go to Step 2: Geometry](#)

[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)