

# ANSYS 12 - Crank - 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

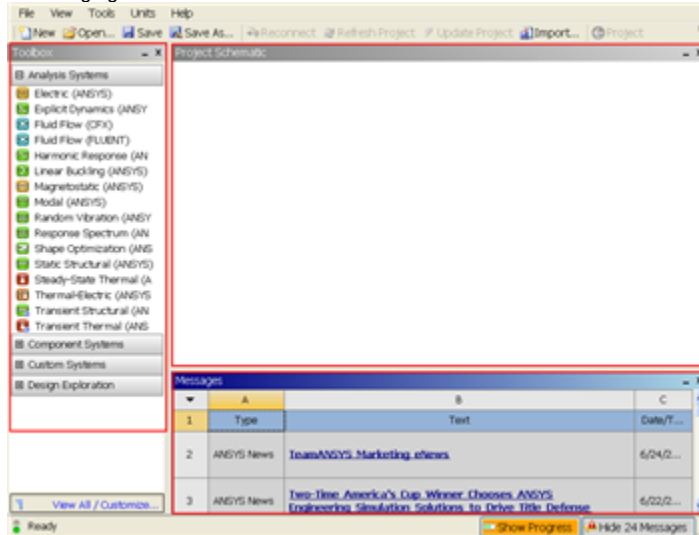
From the problem statement, we know that we are solving static structural problem.

### Start ANSYS Workbench

With the new release of ANSYS 12, there have been a lot of improvement in term of overall flow. We start our simulation by first starting the ANSYS workbench.

**Start > All Programs > ANSYS 12.0 > Workbench**

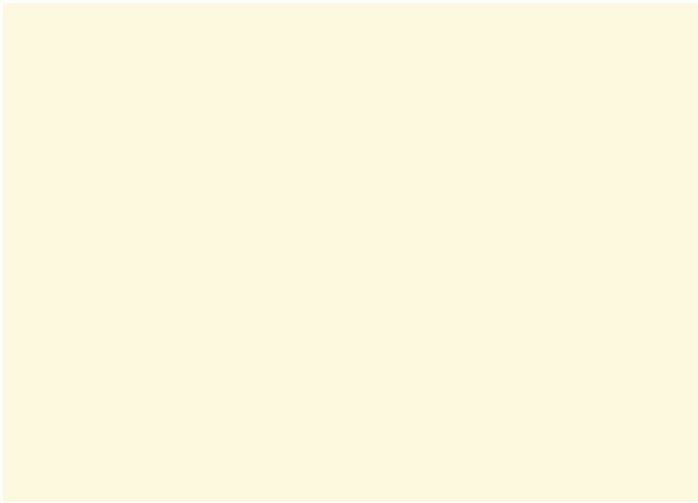
Following figure shows the workbench window.



[Higher Resolution Image](#)

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

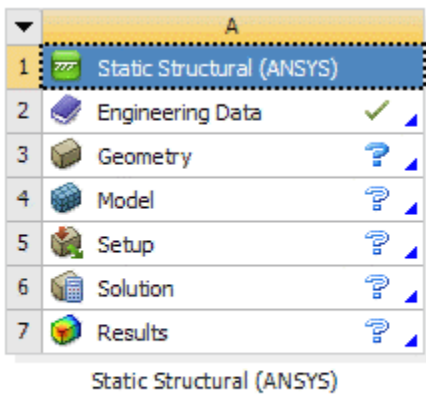


[Higher Resolution Image](#)

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. Rename the project to **Crank**.



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

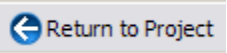
We will first specify the material properties of the crank. The material has an Young's modulus  $E=2.8 \times 10^7$  psi and Poisson's ratio  $\nu=0.3$ .

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**. Under **Isotropic Elasticity**, change **Young's Modulus** and **Poisson's Ratio** to E=2.8e7 psi and =0.3. Remember to change the unit accordingly.

Properties of Chart : Structural Steel				
	A	B	C	D E
1	Property	Value	Unit	 
2	 Density	7850	kg m^-3	
3	 Coefficient of Thermal Expansion			
6	 Isotropic Elasticity			
7	Young's Modulus	2.8E+07	psi	
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		Pa	

[Higher Resolution Image](#)

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](#)