

ANSYS 12 - Beam - Step 5

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 5: Solution

Now that we have set up the boundary conditions, we can actually solve for a solution. Before we do that, let's take a minute to think about what is the post-processing that we are interested in. We are interested in the deflection and bending stress on the beam. We would also like to look at the force and moment reaction at our support A and B. Let's set up those post-processing parameters before we click solve button.

Let's start with inserting Total Deformation.

Outline > Solution (A6) > Insert > Total Deformation

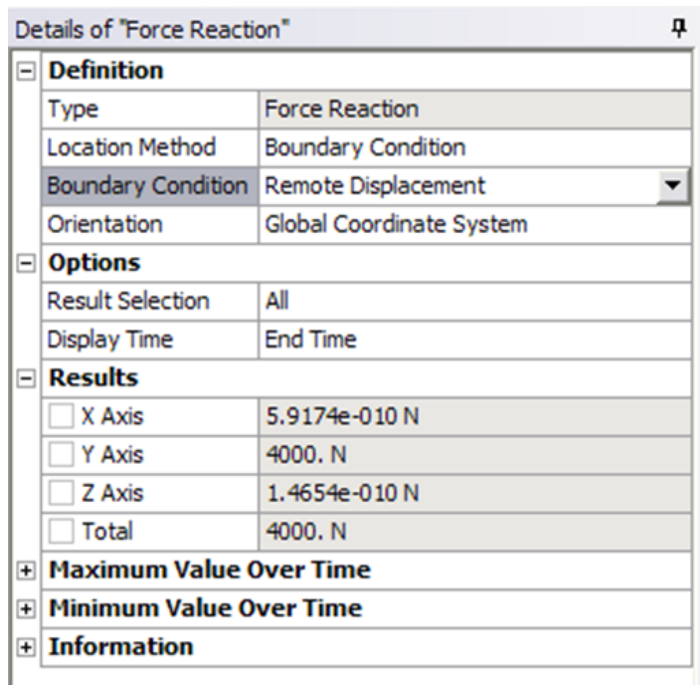
Next let's insert beam tool that will enable us to look at the stresses on the beam.

Outline > Solution (A6) > Insert > Beam Tool > Beam Tool

We would also like to look at the Force Reaction at point A and B.

Outline > Solution (A6) > Insert > Probe > Force Reaction

Select **Remote Displacement** (which is point A) next to **Boundary Condition** under **Details of "Force Reaction"**.










Do the same step for Remote Displacement 2 (point B).

Next we will like to check and see that the moment at point A and B is zero.

Outline > Solution (A6) > Insert > Probe > Moment Reaction

Select **Remote Displacement** (which is point A) next to **Boundary Condition** under **Details of "Moment Reaction"**.

Details of "Moment Reaction" 

	Definition	
	Type	Moment Reaction
	Location Method	Boundary Condition
	Boundary Condition	Remote Displacement
	Orientation	Global Coordinate System
	Summation	Centroid
	Options	
	Result Selection	All
	Display Time	End Time
	Results	
	Maximum Value Over Time	
	Minimum Value Over Time	
	Information	

Do the same step for Remote Displacement 2 (point B).

We are done setting up all the results. Click **Solve** at the top menu to obtain a solution. Wait for a minute for the solution.

[Go to Step 6: Results](#)

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

[Go to all ANSYS Learning Modules](#)