

# ANSYS - Stress due to Gravity

Author: Matt Scott, 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

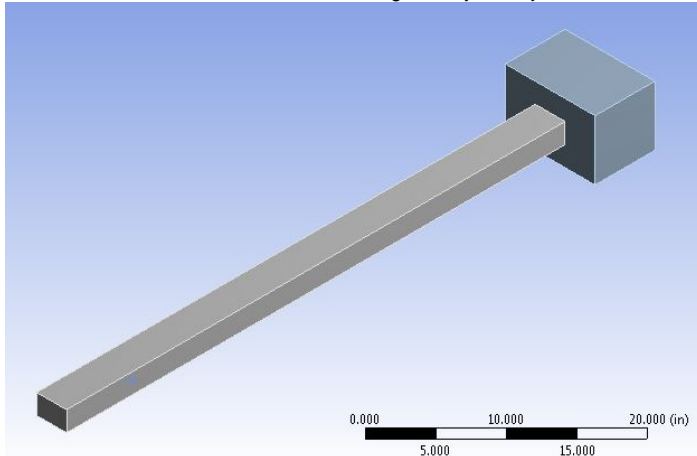
This page has been moved to <https://courses.ansys.com/index.php/courses/stresses-due-to-gravity-using-ansys-mechanical/>  
Click in the link above if you are not automatically redirected in 10 seconds.

## Stress due to Gravity

Created using ANSYS 13.0

### Problem Specification

Consider the geometry in the figure below. The slender bar is 3 inches wide, 2 inches tall, and 50 inches long. The large block on the end of the bar is 9 inches wide, 6 inches tall, and 6 inches long. Both are made of structural steel with a Young's modulus of  $2.90075 \times 10^7$  and a poisson's ratio of .3. The left end of the bar is fixed to a wall, and the geometry is subjected to standard earth gravity. Calculate the maximum equivalent stress in the geometry.



[Go to Step 1: Pre-Analysis & Start-Up](#)

[Go to all ANSYS Learning Modules](#)