

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

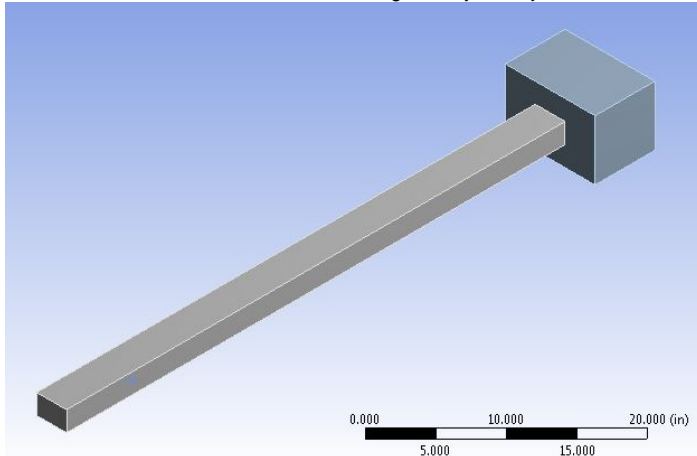
This page has been moved to <https://courses.ansys.com/index.php/courses/stresses-due-to-gravity-using-ansys-mechanical/>
Click in the link above if you are not automatically redirected in 10 seconds.

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 2.90075×10^7 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](#)