Assignment 3: Finite Element Analysis

The objective of this assignment is to familiarize you with the Solid Modeling software, Solid Works, and the Finite Element Analysis software, Cosmos.

Before beginning this exercise, you need to know what the following terms mean:

Boundary conditions, nodes, elements, and mesh

You will also need to provide the following information about the machine component(s) you are modeling:

part(s) geometry and materials

You will build your model in Solid Works and then analyze it in Cosmos.

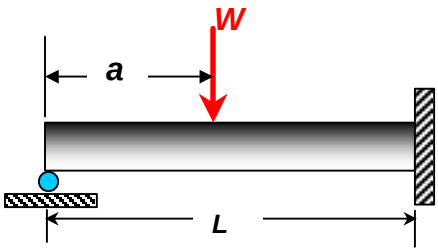
Part 1 of your assignment requires you to first work through Tutorial 3--*Statically Indeterminate Beam*, on our homepage (http://www.eng.iastate.edu/~gkstarns/me325/me325.htm)

Tutorial 3 was written for a slightly different version of Solid Works and Cosmos, but will be nearly the same and will help you get up to speed for part 2 of the assignment.

Part 2

Find the maximum deflection for the beam loaded as indicated and find the location on the beam where the maximum deflection occurs. You are to perform your analysis using Cosmos and you are to confirm that your FEA model is accurate by verifying your solution with either use of superposition or the integration method.

Turn in: printout(s) of your FEA model showing the original and deflected shapes and analytical verification demonstrating the FEA model's integrity (i.e., show that your analytical solution for maximum deflection and its location along the beam closely matches the FEA solution).



cross-section is rectangular (1/2" wide, 1" in height)

The beam is homogeneous and isotropic in compostion.
Bending is planar.

L, the length of the beam is 5 feet.
a, the distance from the left edge to point load W, is 3 feet.
W is 1000 lb_f
The beam is steel (use default values given in Cosmos**)**