

Homework Assignment P-04

Structural Analysis with Femlab

Having completed the structural tutorial, do the following:

1. Create the geometry used in the ANSYS structural tutorial
2. Change the element type to linear quadrilaterals
3. Refine the mesh until you are satisfied
4. Compare the von Mises stress results and maximum deflection with the results you obtained from ANSYS.
Also discuss ease of model generation and your opinion of the script files.

Due: Post results on-line and inform instructor of location by class time Monday.