Due: March 24, 2016 Name _____

HOMEWORK SET # 5
ME/AE 6212 Advanced Finite Element Analysis

Solve the following plate problem using ABAQUS.

Static plate bending of a simply supported isotropic square plate under uniform transverse load. Use four and eight node Quad elements. The plate dimensions are: 10 in x 10.in x 0.1 in.

Compare the center deflection with the exact results presented in Table 12.5.1 (Reddy's text) for linear analysis.

Use a uniformly distributed load of 500 psi.

Material properties for ABAQUS input: E=30 x 10^6 psi, Poisson's ratio=0.25.

Need a summary of results and sample output. (20 points)

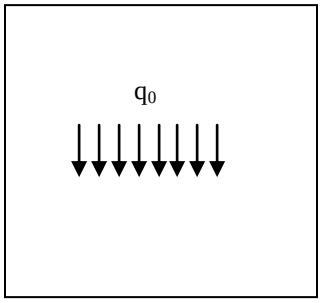
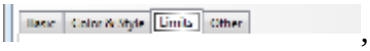


Plate under bending load

Hint: To get the maximum deflection from ABAQUS plots, plot the displacement in the thickness direction (e.g. U3). In the contour plot options,  ,

Go to Limits  ,

and then mark "specify location" for " max" 

This specifies the magnitude of maximum displacement and its location (at the center). Ignore the negative sign for U3.