The objective of this study is to determine the allowable load of a lever arm and the maximum deflection that occurs under that load. The maximum von Mises stress should not exceed the yield stress of the lever arm material, and this criterion should be used to determine the allowable magnitude of the uniformly distributed load $P$.

We consider a lever arm under static loading, as shown in Figure 1. The lever arm is made of steel with a mass density of 0.283 lbm/in$^3$, a Young’s Modulus of 30E6 psi, a Poisson’s ratio of 0.3, and yield stress of 52.0 ksi. The lever arm has a circular cutout of radius $R_1=1.0$ inches, and center location specified by $X=2.5$ inches and $Y=-2.5$ inches. The lever arm has a fillet $R_2=1.0$ inches, and the height of the primary beam $H = 5$ inches. The uniform thickness of the part (in the Z direction) is 0.1 inch. Also shown in Figure 1 are the applied load and boundary conditions. The lever arm is clamped along its left edge and exposed to a uniformly distributed load on its top edge.

Your assignment is to perform a finite element analysis of this structure in Abaqus to determine the allowable load $P$ and the associated deflection. Your investigation must include a detailed mesh convergence study and a validation study. A full report is required.

The full report is due at noon on Thursday, May 5, 2011.

Please put the report in Dr. Averill’s faculty mailbox or under his office door by the due date.