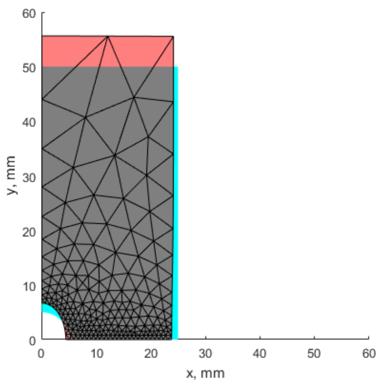
Assignment 6 - 3D Tet Hole-in-Plate

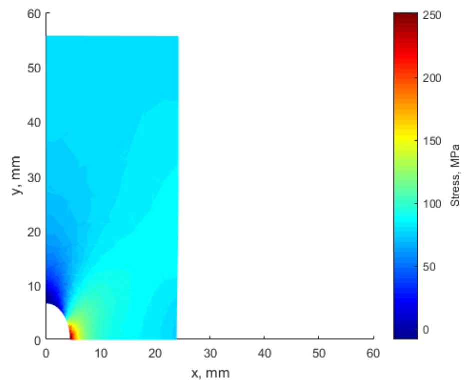
Kalin Gibbons

November 23, 2016

1 Problem Statement



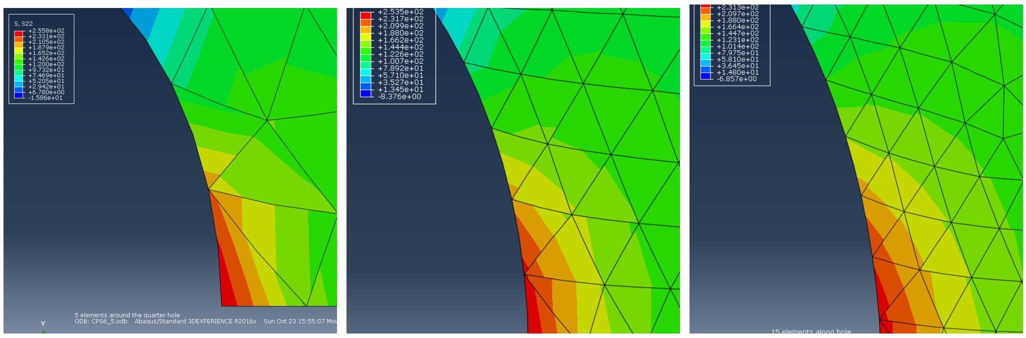
(a) Deformation plot



(b) Stress - σ_{22}

Figure 1.1: Matlab plots of deformation and stress for the quarter plate-with-hole at a nominal mesh density

Perform a convergence study with 6-noded tri elements, using at least 3 mesh densities. For each mesh generated in your mesh convergence study, re-run with 3-noded tri elements. Write a finite element program in Matlab to solve for stresses in your nominal mesh with 6-noded elements. Your program must include calculation of the displacement solution (using the 3-point rule), and calculation of stresses directly at the nodes. Plot the results of stresses in the y-direction. Determine the stress concentration factor.



(a) 5 elements (b) 10 elements (c) 15 elements

Figure 2.1: Abaqus plots showing discontinuities at element borders during mesh refinement. Labels refer to the number of elements around the hole

Table 2.1: Calculated top edge displacement and stress concentration factors for various meshes

Element Seed	**Top Displacement (μm)**	**Concentration Factor**
5	703.093	2.55
10	703.165	2.53
15	703.145	2.53

2 Convergence Study

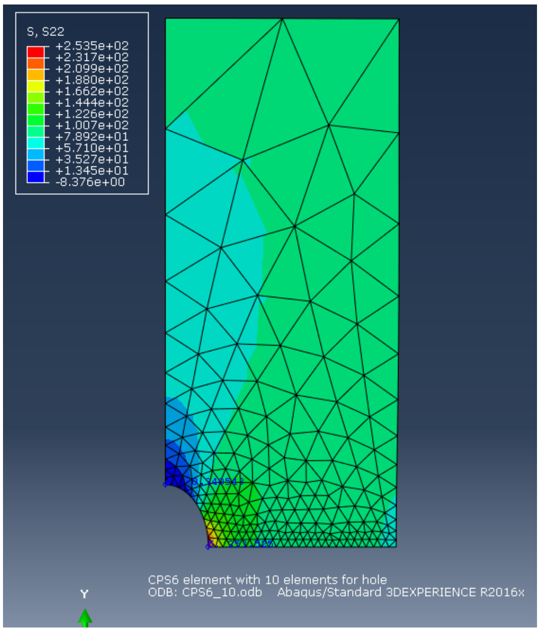
Our mesh converged with an edge seed of 10 elements around the quarter hole, and 2 elements across the top half of the plate. Stress discontinuities across each element may be seen in Fig. 2.1 for each mesh density. The discontinuities are increasingly less severe with each increase in density. While the stress bands appeared much smoother with the 15 element seed, the stress concentration factor remained constant at 2.53 for both the 10- and 15- element seed, which was close to our nominal value of 2.54. Numeric comparison of the 3 mesh densities may be seen in Table 2.1.

Table 3.1: Element performance comparison between constant- and linear- strain triangles. Stress and displacement given in MPa and μm, respectively

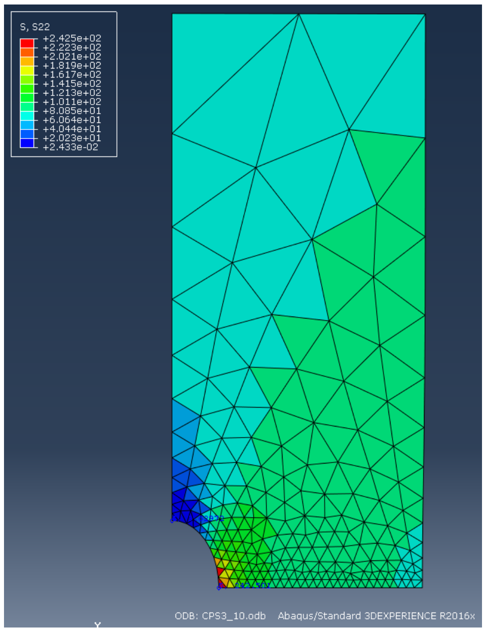
Seed	Constant				Linear			
	$\sigma_{2,bot}$	$\sigma_{2,side}$	$U_{2,hole}$	$U_{2,top}$	$\sigma_{2,bot}$	$\sigma_{2,side}$	$U_{2,hole}$	$U_{2,top}$
5	223.479	8.420	195.050	700.126	255.773	−0.9015	209.282	703.093
10	242.501	3.62972	203.129	702.077	253.525	−0.2405	209.787	703.165
15	247.477	1.30168	205.545	702.720	252.979	−0.1603	209.821	703.145

3 Element Performance

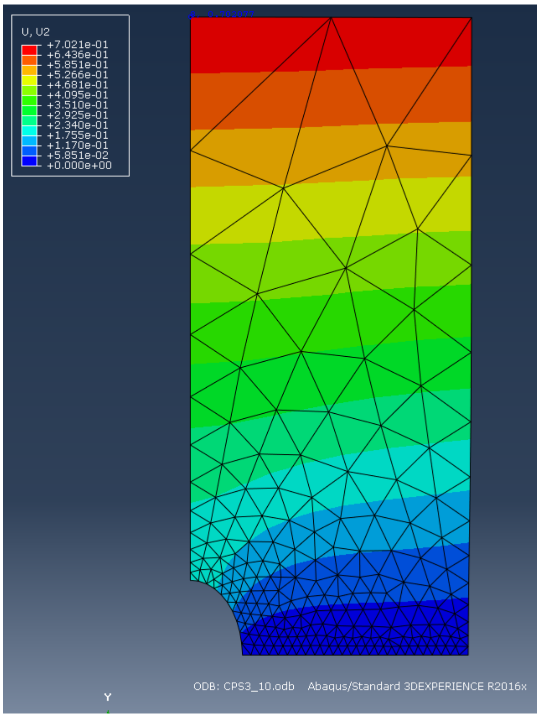
Numerical values for each type of element may be seen in Table 3.1. The constant strain elements consistently under-calculated the maximum stress around the hole, and over-calculated the low stress at the top of the hole. Displacement of the top of the plate was under-calculated as well, by ranges of 0.425 to 2.967 μm. The stress and displacement plots for the 10 element seed for both constant- and linear- strain triangles may be seen in Fig.3.1. The CST elements differ along the top left region of the quarter plate, and the stress plot shows greater discontinuities across the edges of the CST elements. Any bending in this problem was through the eccentric loading of the hole caused by the distributed load across the top of the plate, and we should be mindful of this in future problems that do not appear to be under only axial loading conditions at first look.



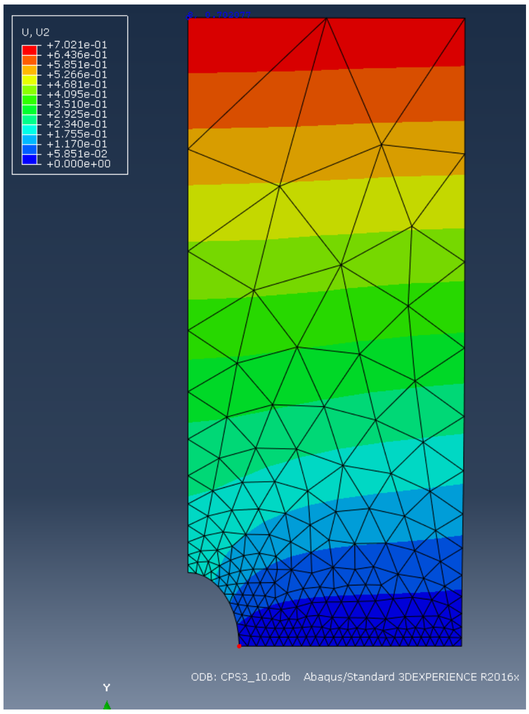
(a) LST σ_2



(b) CST σ_2



(c) LST U_2



(d) CST U_2

Figure 3.1: Linear- and constant- strain triangle stress and displacement plot comparisons from Abaqus

Table 4.1: Abaqus stress (MPa) values for the critical element with 10 element seed

Node	14	5	15	258	259	260
S11	3.88	0.766	27.4	2.32	14.1	15.7
S22	243	253	178	248	216	210
S12	−31.5	−1.06	2.85	−16.3	0.893	−14.3

Table 4.2: MATLAB stress (MPa) values for the critical element with 10 element seed

Node	14	5	15	258	259	260
S11	2.212	0.289	26.6	2.46	13.5	16.1
S22	239.1	251.6	177	249	214	210
S12	−32.5	−1.04	2.59	−16.2	0.396	−13.9

4 Comparison to MATLAB

The two solution methods provided similar results for stress in the x, y, and xy directions. The greatest difference in values was 3.8403 MPa, and the smallest difference was 19.5837 kPa. This may possibly be explained by Abaqus' use of extrapolation of nodal stress values from the integration points of the elements. The individual values for each stress may be seen in Tables 4.1,4.2, and a plot of the critical stress region may be seen in Fig.4.1. Some discontinuities are seen across the element edges, but the stress is relatively continuous.

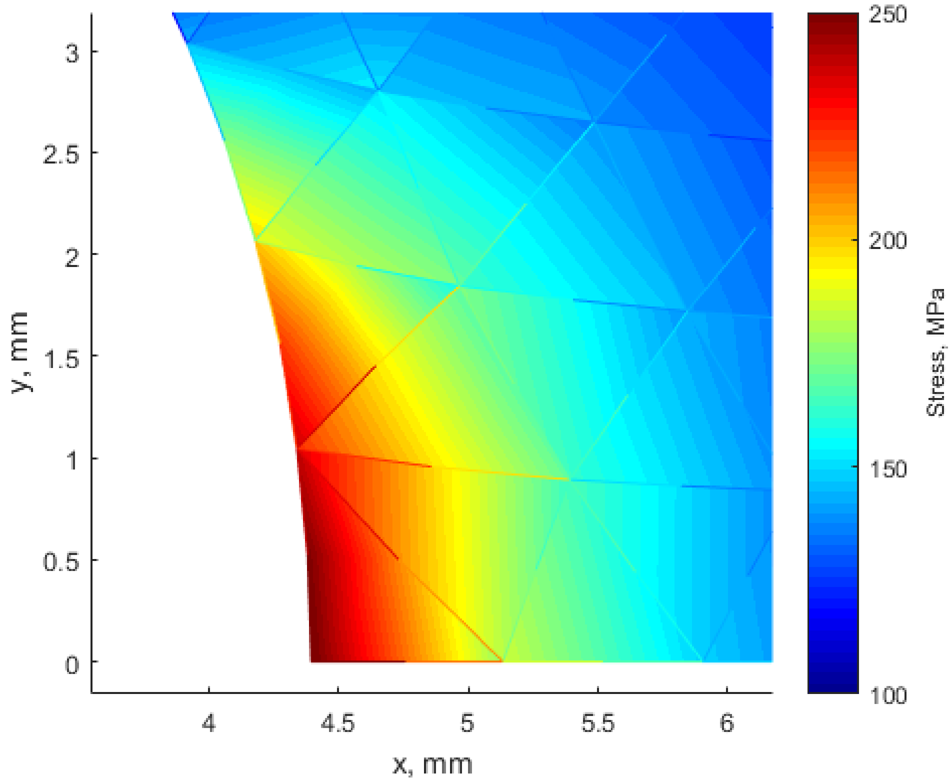


Figure 4.1: MATLAB plot of critical region around the hole of the plate for σ_y