Laboratory Assignment No. 1.

Objectives:
Learn to set up and solve problems using StressCheck\textsuperscript{1} or some other finite element analysis (FEA) software product. Learn to verify information computed from finite element solutions.

Problem Statement.
Analyze the notched rectangular plate loaded by a shear force $V$ and bending moments $M_1$, $M_2$ as shown in the figure. The material is 6061-T6 aluminum.

1. Define the solution domain. Define all dimensions as parameters. For initial values use $a = 180$ mm, $b = 30$ mm, $c = 45$ mm, $d = 65$ mm, $h = 24$ mm, $R = 3$ mm, thickness 4 mm. Impose the necessary restrictions on the parameters (for example, $c + d + 2R < a$). Note that $c$, $t$, $r$ are among the reserved parameter names in StressCheck, so use other parameter names (e.g., YM for Young’s modulus, TH for thickness, etc.)

2. Construct a finite element mesh manually. Observe how the mesh changes as you change the dimensions.

3. Assign the thickness and the material properties.

4. Define formulae for the normal and shearing tractions on boundary segments BC and DA. Define $V$ and $M_1$ as the independent load parameters\textsuperscript{2}. Use $V = 250$ N and $M_1 = 10 \times 10^3$ Nmm. Hint: Define a local coordinate system, the $x$-axis of which is normal to BC and DA and passes through the center of the boundary segments.

$$T_n = -\frac{My}{I} \quad T_t = -\frac{VQ}{It}$$

where $T_n$, $T_t$ are the normal and tangential components of the traction vector and

$$I = \frac{b^3t}{12} \quad Q = Q(y) = \frac{1}{8}(b^2 - 4y^2)t.$$

Perform an equilibrium check after you entered the data.

\textsuperscript{1}StressCheck is a trademark of Engineering Software Research and Development, Inc.

\textsuperscript{2}Note that $M_2$ is not an independent parameter. From equilibrium: $M_2 = aV + M_1$. 

---

[Diagram of the notched rectangular plate with labels for dimensions and loadings]

---
5. Impose rigid body constraints.

6. Obtain finite element solutions for $p = 1, 2, \ldots, 8$. First use the trunk space then switch to the product space.

7. Perform a design study. Allow either $R, c$ or $h$ to vary between reasonable limits and plot the maximum von Mises stress as a function of the parameter of your choice.

Documentation.

You can access the manuals of StressCheck by selecting Help/MasterGuide from the main menu. Go to Bookmarks to view the main headings. The keywords are found in the Index. By clicking on a section number in the Index you will open the corresponding page of the manual.

Report.

Describe what you have done in a brief and informal report. The report must include a figure showing the finite element mesh, the estimated error in energy norm, a contour plot of the von Mises stress and convergence plots for the maximum and minimum normal stresses\(^3\). Answer the following questions:

1. What do you consider to be the most important thing that you learned while working on this assignment?
2. What difficulties (if any) did you encounter in learning the use of the FEA program?
3. Approximately how much time did you spend on this assignment?
4. Why is the potential energy negative?
5. Does the stress distribution depend on the material properties? - Explain.
6. Does the stress distribution depend on whether plane stress or plane strain condition is assumed? Explain.

Extra credit.

You may use any FEA software product to satisfy the requirements of this assignment. You will receive extra credit if you complete at least the first five items of the Problem Statement using more than one FEA software product.

If you use StressCheck you can still earn extra credit by performing any or all of the following and include a documentation of your extra credit work in your report.

1. Select “Extrude” and perform the solution again. Make sure that the plate is properly constrained against rigid body motion in 3D space.
2. Explore the auto-meshing capabilities of StressCheck.
3. Explore the handbook capabilities of StressCheck.

Due date.

Your report is due on October 21, 2009.

\(^3\)To find the maximum tensile and maximum compressive stress in planar analysis, one has to find the maximum value of the first principal stress and the minimum value of the second principal stress.