Project#2 Analysis and Design of Bracket

A bracket shown in Figure 1 has following properties: Young's modulus E = 210GPa, Poisson's ratio ν = 0.29, design ρ = 7850kg/m^3, thickness = 3mm, and maximum allowable stress = 800MPa. A horizontal force F_x = 10,000 N is applied at the center of upper hole and two bottom holes are fixed to the ground.

Task 1: Geometry modeling (30pts). Using Abaqus, make a sketch with appropriate dimensions and constraints. Once all constraints are properly applied, geometry curves will change color to green. Your model should be able to be updated properly when design variables are modified. For boundary conditions, the circumference of the two bottom holes can be fixed (u = v = 0). Force has to be applied at the center of the upper hole. First create a reference point at the center, and define MPC constraint with link-type between the reference point and the circumference of the hole. Then, apply the horizontal load that the reference point. In the report, show your constrained geometry with dimensions.

Task 2: Convergence study (20pts). Convergence study is to show that finite element solution converges to a value as more and more elements are used. Use 4-node quadrilateral elements (without reduced integration), perform a convergence study of x-displacement at the load application point as a function of the number of nodes. Since the current version of Abaqus SE only allows up to 1,000 nodes, it would be a good idea to change the number of nodes by approximately 400, 600, 800, and 1,000. Plot displacement versus the number of nodes and show is any convergence can be observed. Discuss why or why not observe convergence. In the report, show the graph of number of elements versus tip x-displacement.

Task 3: Element comparison study (20pts). In the class, we learned that different elements perform differently. The purpose of the element comparison study is to compare analysis results when different elements are used. Use four different element types (3-node triangle, 4-node quadrilateral, 6-node triangle, 8-node quadrilateral) and compare difference in x-displacement at the load-application point and maximum von Mises stress. Use the same global element size such that all element sizes are similar. Discuss the results in terms of accuracy and why certain elements provide a good results, but others are not.

Task 4: Minimizing the mass by changing 9 design variables (20pts). The initial design has some margin of safety for maximum allowable stress. The goal of Task 4 is to change the 9 design variables to reduce the mass as much as possible, while the maximum von Mises stress is less than the maximum allowable stress. First, select the best element type and element size from Task 2 and Task 3. When you change designs, do not randomly change them. You have to use your own design strategy (based on your knowledge from Mechanics of Materials) and explain clearly in the report. Also, design change should maintain the current topology of the geometry. Compare the difference between the initial and final designs in terms of the mass and maximum stress. Points will be weighted based on the mass and accuracy of calculation.

Task 5: Reporting (10pts). Formal report is required, including title, summary, introduction, approach, results, discussion, appendix (input file, programs), and/or references. Report must be self-explanatory; define all terms that you use and explain clearly what you did. The most important criterion for reporting is clarity and self-explanatory. Report should be less than 10 pages, but it must contain all necessary information. Submit report (Word or PDF file, Max 10 pages), CAE and ODB files at minimum mass design. The following penalty or extra credit will be given depending on submission date.

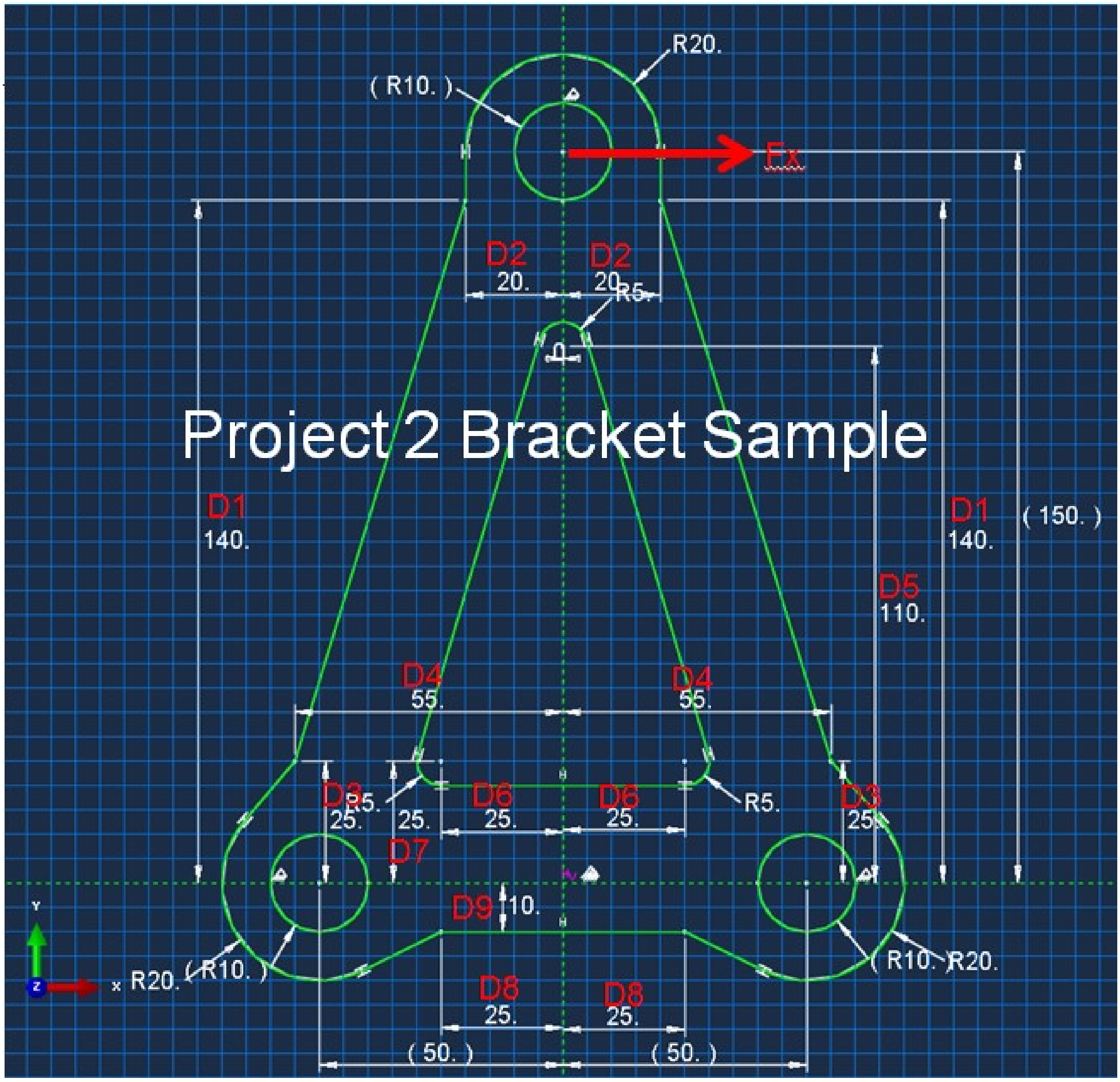

Figure 1. Geometry of a bracket (unit mm)