Department of Mechanical Engineering
Mechanics and Materials (MCEN30017)
Part 3: Finite Element Analysis (FEA)
Semester 1, 2023

Assignment

Objective:

This assignment aims to evaluate students' ability to use an analytical FEA approach to solve 1D/2D structural problems (see examples in lecture notes), and utilise a commercial FEA package to assess the structural integrity of a 3D engineering system. Students will use SolidWorks computer software to complete this assignment.

Assessment:

This assignment is worth **25% of your total mark**. The report is to be completed by each student individually and submitted online by **5 pm May 26 (Friday, week 12), 2023**. The report should not exceed 20 pages, including figures, tables, and appendices.

Section 1: FEA analytical approach

Q1: Figure 1 shows a plane truss composed of three bar elements subjected to downward force, *P*, applied at node 1. Determine the displacements at node 1 and the stresses in each element.
Let Young's modulus E = 200GPa & cross-section area A = $6 \times 10^{-4} m^2$ for all elements, and P = 50kN. (15 marks)

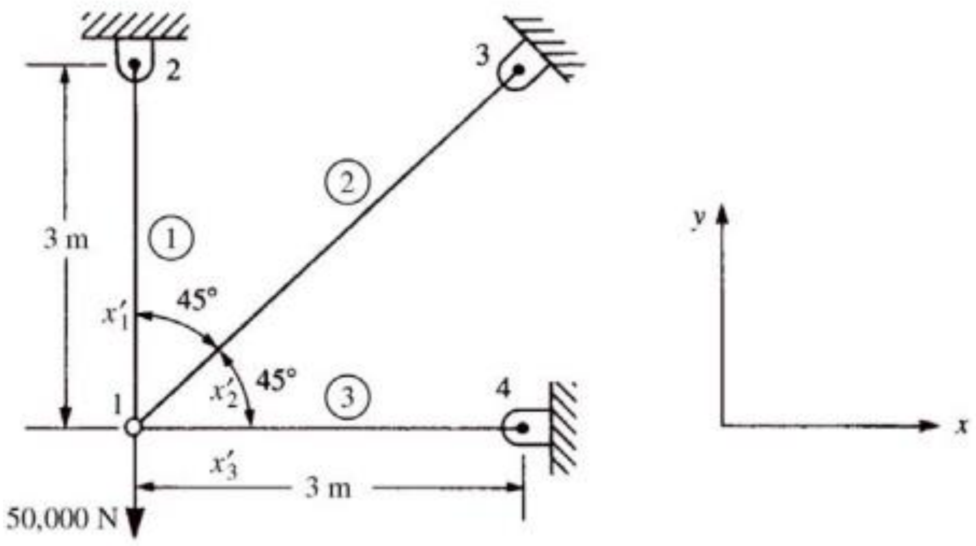


Figure 1

Q2: For the plane truss shown in Figure 2, determine the nodal displacements, the element forces and stresses and support reactions. All elements have Young's modulus E=700GPa and cross-section area A = $3 \times 10^{-4} m^2$. Verify force equilibrium at nodes 2 and 4. Use symmetry in your model. (15 Marks)

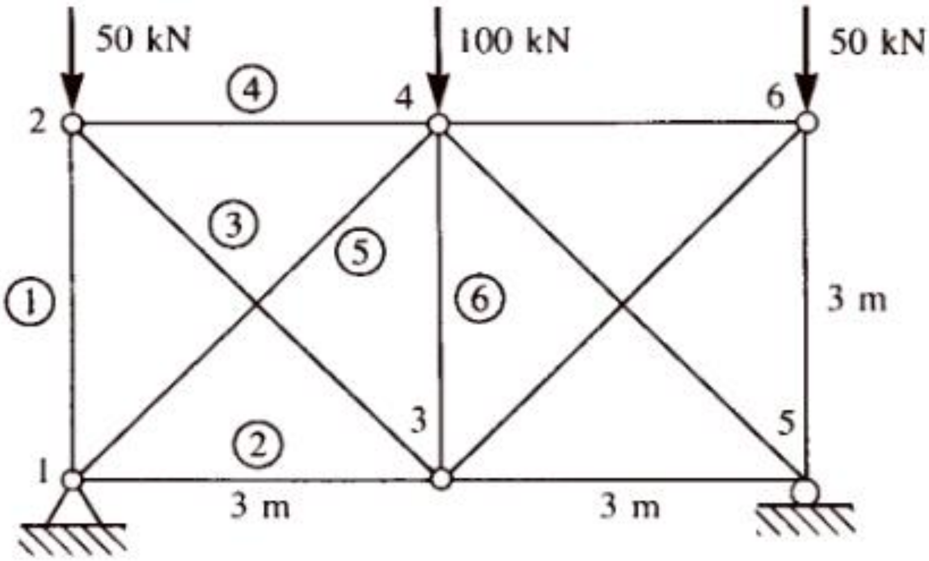


Figure 2

Section 2: Computer-based stress and strain analysis

Introduction:

A nursery is considering a new design for a plant stand as depicted in Figure 3. This system is required to exhibit an excellent mechanical performance against holding plants and heavy pots up to 50 [kg]. The pots, including some heavy items are placed on top of each shelf. These regions are illustrated in Figure 3 as A, B, and C. The main beams are constructed from a 30 mm × 20 mm rectangular profiles, with 1.5 mm thickness. Before analysis, some pre-assumptions are made as follows:

1) The weight of the stand can be neglected and thereby excluded from the analysis.
2) The tool table is made of 2018 Aluminum Alloy (yield strength = 317.1 MPa).
3) The table is assumed to be fixed to the ground.
4) Shelves A, B, C1, and C2 are exposed to 30 kg, 40 kg, 50 kg, and 40 kg loads, respectively.
5) The upper holder shaft and profile cross section are also exposed to a total force of 10 kg (note the direction of the applied loads in Figure 3).
6) The loads are <u>evenly</u> distributed on each surface. Assume g=10 m/s^2

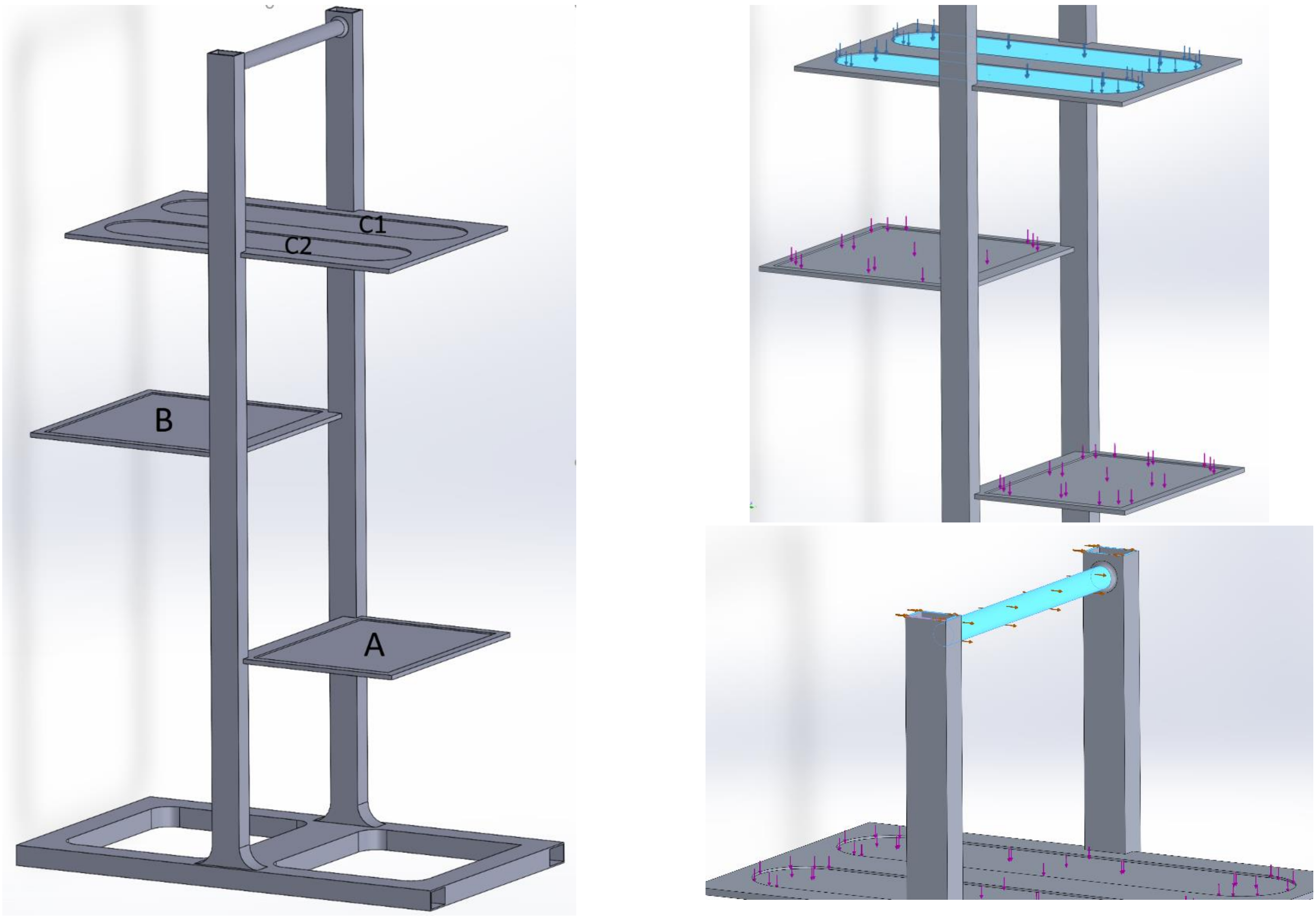


Figure 3. Schematic of a plant stand and applied loads.

Part 1:

Structure your report such that you cover all the questions below.

1- Construct a model for the tool table in SolidWorks, and provide a figure showing the final mesh, loads and constraints. Please present your model as a third-angle projection with all relevant dimensions. (5 marks).
Hint: For ease of modelling and subsequent optimization, it is recommended to define global variables for parameters in the design: width, height, and thickness of the beam profile as well as fillet radius (Figure 4). This can be done in SolidWorks by clicking on Menu>Tools>Equations>Add Global Variables. For sketching, 3D featuring, and the optimization analysis you should use this predefined Global Variables for fillet radius, width, height, and thickness of the beam profile.

2- State and justify how you have modelled the loads and constraints, along with any assumptions you have made regarding this. Is it physical to assume the fixed boundary condition for the supported legs? What are the proper boundary conditions for this problem? (10 marks)

3- Perform a mesh sensitivity analysis to find an appropriate global and local mesh size. Justify the reason behind choosing the local mesh location. Present a figure showing average stress vs. mesh size, as well as a figure showing computation time vs. mesh size. (10 marks)
Hint: You can choose the convergence of total average von Mises stress for the global mesh size sensitivity analysis and an average of von Mises stress at the location of local mesh for the local mesh sensitivity analysis.

4- Provide a von Mises stress contour plot of the table. Identify the location and magnitude of the region of highest stress. (5 marks).

5- FEA is a powerful engineering design tool, and programs such as SolidWorks facilitate the design of complex three-dimensional structures, bypassing arduous calculation methods of the past. These tools, however, are not infallible; rather, they suffer from many of the same issues the calculation method they replaced faced. Elaborate on possible errors in your FEA analysis (mention three at least), and propose methods for verifying your FEA results (Only propose) (10 marks)

Part 2:

A second design is required for the tool rack to withstand a load that is 50% higher than the one used in the initial design. The goal is to minimise the mass, keeping the maximum von Mises stress less than its yield stress and the maximum resultant displacement (URES) less than 16 mm.

6- How does an independent increase in profile thickness and fillet radius change the maximum stress and displacement in the body? Which parameter has a greater effect? Present a plot demonstrating the effect. (10 marks)

7- Find the optimal geometry by changing both the beam profile thickness (for all beams) and the fillet radius according to the information in the table below. Plot the results on an URES displacement vs. beam profile thickness graph, with separate lines for each beam profile thickness value. Present the optimal geometry in a table, including the maximum stress and displacement. (15 marks)

Table.1. Fillet and beam thickness sizes for optimal design.

	Minimum	Step Size	Maximum
Beam Profile Thickness	1 [*mm*]	1 [*mm*]	5 [*mm*]
Fillet Radius	5 [*mm*]	5 [*mm*]	25 [*mm*]

8- Provide a von-Mises stress contour plot of the optimized table. Identify the location and magnitude of the region of the highest stress. (5 marks)

`

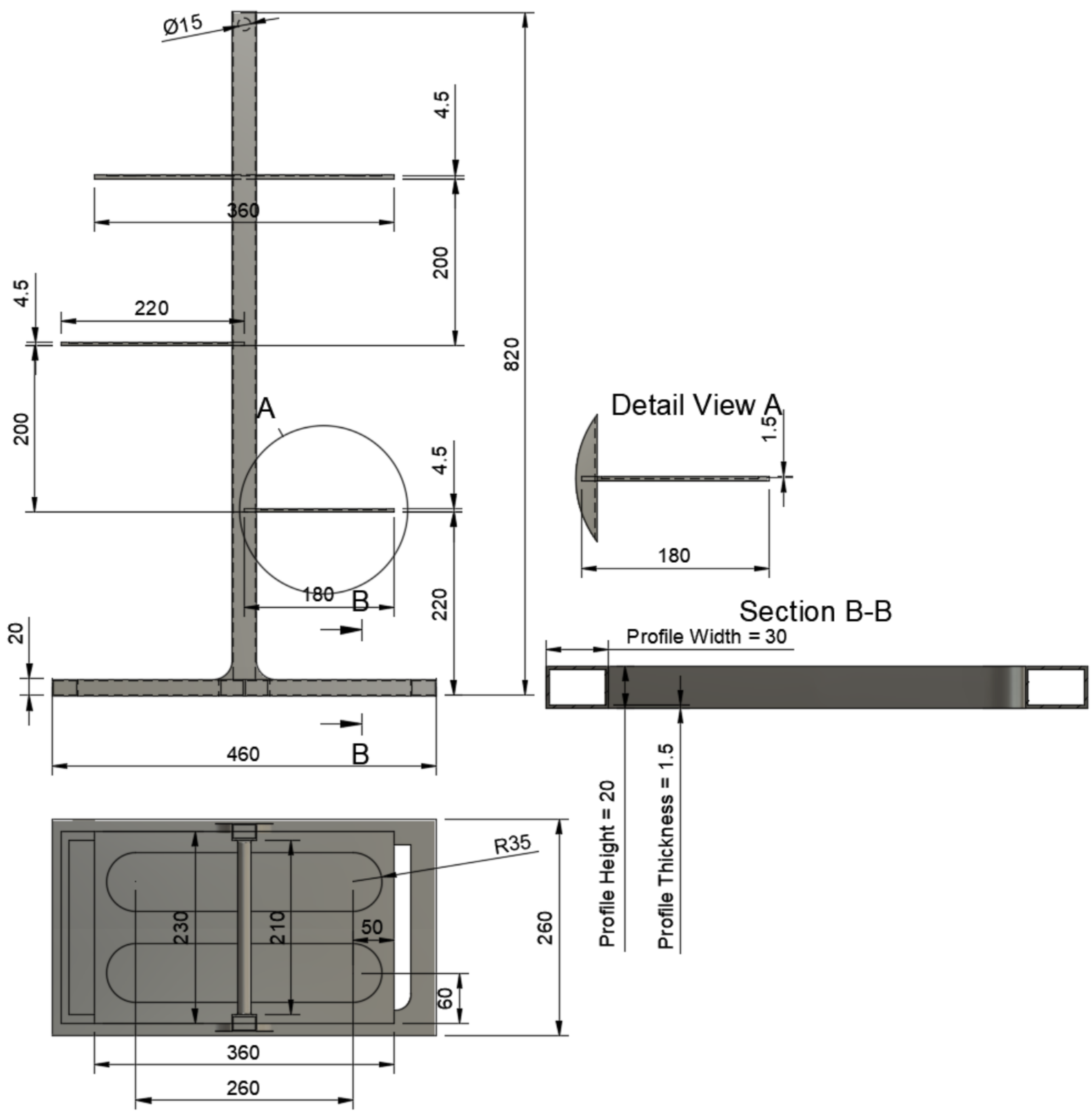
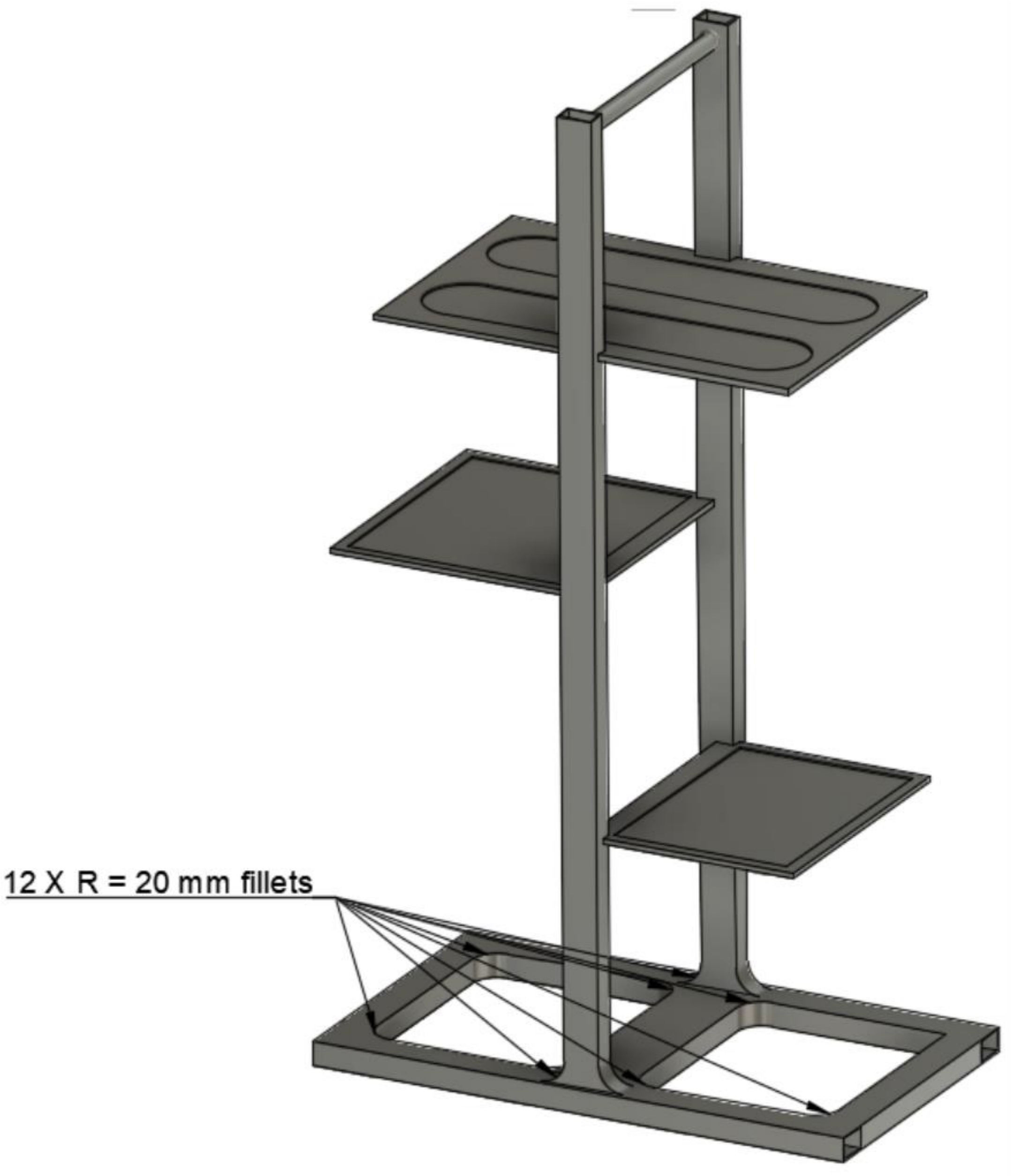


Figure 4. The 3D Isometric view, third-angle projections, and geometrical dimensions of the proposed tools table. All the main beams (excluding the shelves) are made of 30 mm × 20 mm rectangular profiles (1.5 mm thickness), as shown. Each shelf is **3 × thickness of the profile** (in this drawing because the thickness is set as 1.5 mm hence the shelves' height are 4.5 mm)



NOTE: All dimensions are in mm