1 Replication of results

This chapter describes the step-by-step procedure to do topology optimization of a chair as shown in Fig. A.1 using Ansys topology optimization 2020 R1. The design domain for chair is fixed at bottom four corners loads are applied as shown in fig. 1 . The optimization problem is to minimize the compliance along with a mass/volume constraint. The design domain and non-design domain are differentiated with grey and red colors respectively as shown in fig. 1

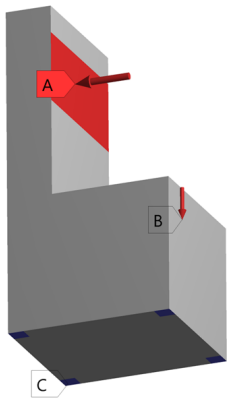

Figure 1: The initial CAD model of chair for level-set topology optimizationand lattice optimization. Legends A,B depict the forces at the seating area and spine resting part in the chair. These are considered non-design domains and are shown in red. The four bottom corners (blue color) are support members and they are also considered part of non-design domain. The grey areas represent the areas for optimization.

1. Drag and drop static structural from Analysis Systems.
2. Enter the material properties by double clicking Engineering Data.
3. Import the CAD geometry in Geometry by right clicking geometry → Import Geometry → Browse and choose the CAD file-Initial domain.step
4. Double click Model → right click on Mesh under Project → choose the element size and mesh the geometry.
5. Right click Static Structural under Project → Insert → Fixed Support → select the fully restrained faces in Geometry under Scope tab. The four bottom rectangles in the model are selected.
6. Right click Static Structural under Project → Insert → Pressure → Select the traction faces in Scope tab and provide pressure magnitude in Magnitude under definition tab.
7. Right click Solution → Insert → Stress → Equivalent (von-Mises).

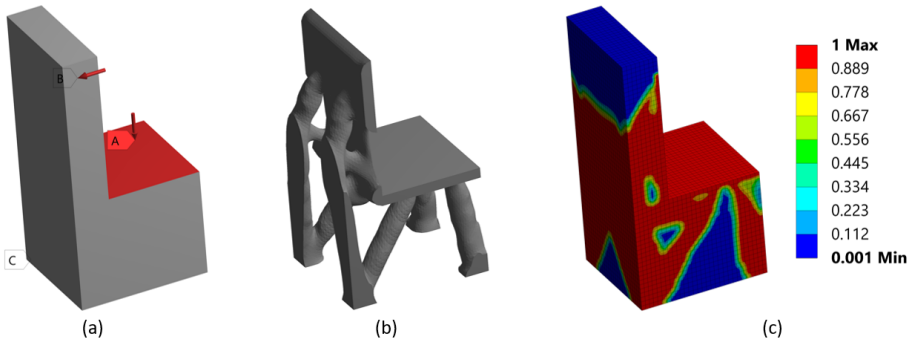

Figure 2: Optimization results. (a) Initial design domain for optimization. (b) Organic shape of chair obtained after Level set based topology optimization. (c) Density contours for homogenization based lattice topology optimization with cubic lattice.

8. Right click Solution → Insert → Displacement → Total → Solve under Home / Solution Tab.

9. Goto Hometab → Select Topology Optimization from Analysis Systems → drag and drop Topology Optimization onto Solution of Static structural tab in the Project Schematic.

10. Double click Setup under Topology optimization project schematic. Select Optimization region under Topology optimization tab → Select the non-design domain under Exclusion Region → Boundary Condition → All Boundary Conditions → goto Optimization Type and select Topology Optimization- Level Set Based.

11. For homogenization based lattice TO, double click Setup under Topology optimization project schematic. Select Optimization region under Topology optimization tab → Select the non-design domain under Exclusion Region → Boundary Condition → All Boundary Conditions → goto Optimization Type and select Topology Optimization- Lattice optimization → Minimum density select 0.001 → Maximum density select 1 → Lattice cell size → Choose your lattice cell size for your application(here we have chosen 4 mm).

12. Select Objective → Response type → Compliance.

13. Select Response Constraint → Definition → Response → Volume/Mass → Percentage to Retain → Select the requsite value → Solve.

14. To view Optimized geometry, select Topology Density. To get smoothened results right click Topology Density → Insert → Smoothing and to view the homogenized optimal densities → Select Lattice density in the Solution section.