1 Replication of results

This chapter describes the step-by-step procedure to do topology optimization of a chair using Ansys topology optimization 2020 R1. An initial design domain of the chair, as shown in fig.1, is applied with seating and spine resting surface pressure loads and is fixed in the bottom four squares, i.e. the contact area with the ground. The optimization problem is to minimize compliance along with a mass/volume constraint in two scales, respectively macro scale for optimizing the geometry and a micro scale for optimizing the material microstructure.

The boundary conditions are considered non-design domains; hence, the level set-based topology optimization method is used to optimize the geometry of the initial design domain. Further, the resultant optimized geometry is fed to micro-scale optimization using homogenization-based lattice optimization. The design and non-design domains are differentiated with grey and red/blue colours, respectively, as shown in fig.1
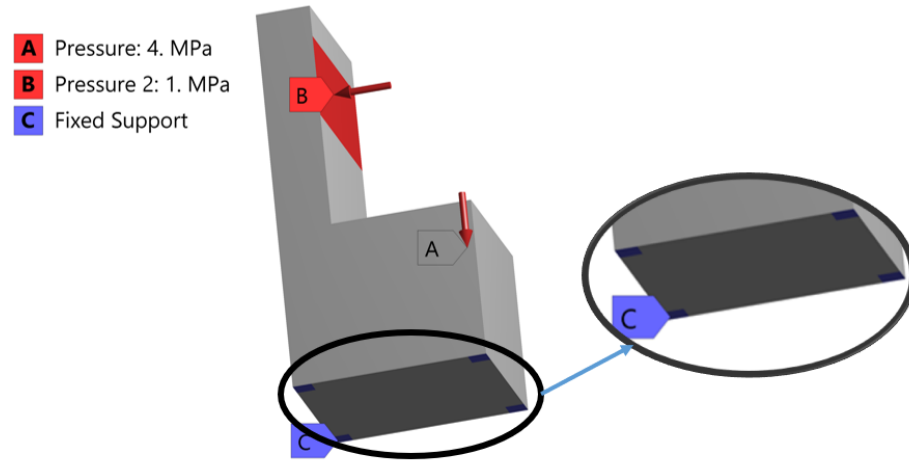

Figure 1: The initial CAD model of the chair for level-set topology optimization and lattice optimization. Legends A and B depict the forces at the seating area and spine resting part in the chair. These are considered non-design domains and are shown in red. The four bottom corners (blue colour) are support members and are also considered part of the non-design domain. The support members are fully restrained and is represented as Fixed Support, as shown in the legend. The grey areas represent the areas for optimization.
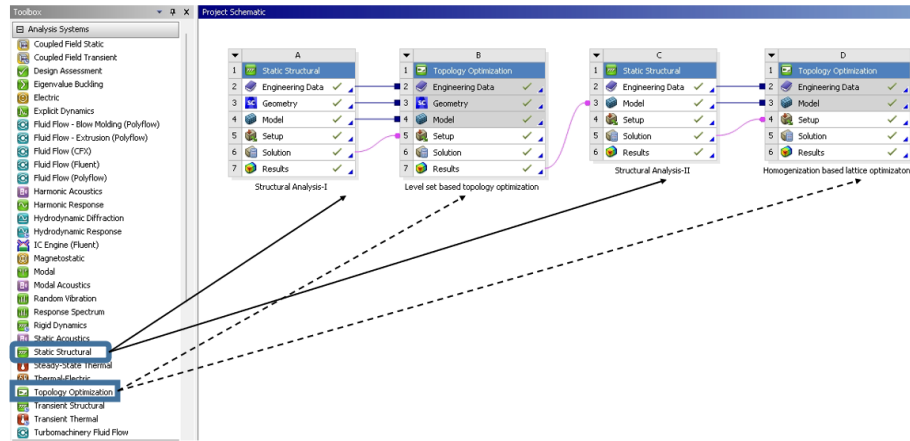
Figure 2: Project schematic for the problem

1. Drag and drop static structural from Analysis Systems and name it Structural Analysis-I

2. Enter the material properties by double-clicking Engineering Data. The parameters such as Density, Isotropic Elasticity can be dragged and dropped to create any customized material. In this work, we use the default material, Structural Steel.

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 → click on Mesh under Project → choose the Element Size and mesh the geometry. Here we chose a value of 0.5 mm.

5. Right click Static Structural under Project → Insert → Fixed Support → select the fully restrained faces (the bottom four squares shown in 1) in Geometry under Scope Geometry. The four bottom squares in the model are selected.

6. Right-click Static Structural under Project → Insert → Pressure → Select the loading surfaces in the Geometry tab under the scope section and provide pressure magnitude in the Magnitude tab under the definition tab. Here we choose 4 MPa and 1 MPa for the sitting and spine resting areas, respectively.

7. Right click Solution → Insert → Stress → Equivalent (von-Mises).

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

9. The result contours comprising of Equivalent stress (von-Mises) and Total deformation can be visualized after clicking the respective tabs under Solution. Other results, such as Principal stress, Principal strain, can be visualized similarly.

10. Note if the user wishes to analyze a multi-loading scenario, the procedure is simple. Click Analysis settings, and in the Step Controls, the Number Of Steps, Current Step Number, and Step End Time can be chosen equal to the number of loading cases. Enter the load values corresponding to each load case in the force type selected (here Pressure) corresponding to each time step. The results for each of the loading scenarios can be visualized by selecting the appropriate Display Time corresponding to the load step.

11. To perform the topology optimization, go to the project schematic page as shown in fig 2. Goto Project schematic → Select Topology Optimization from Analysis Systems → drag and drop Topology Optimization onto Solution of Static structural tab selected previously in the Project Schematic.

12. 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.

13. Select Objective → Response type → Compliance in the work sheet shown in the right.

14. For analyzing the weighted multi-load case, click on the worksheet → Add. Select the appropriate Step numbers corresponding to the loading cases in Start Step and End Step. The corresponding weights for optimization can be appended under the Weight column.

15. Select Response Constraint → Definition → Response → Volume → Percentage to Retain → Select the requisite value (Here a retention volume of 20 % is chosen) → Solve under the home section in the top.

16. To view Optimized geometry, select Topology Density. To get smoothened results right, click Topology Density → Insert → Smoothing. Select Yes to export the model.

17. To analyze the optimized geometry, right-click results → Transfer to Design Validation System. Open the New static structural model that is created. Update the project before proceeding further by selecting Update Project in the top left of the Project Schematic section.

18. The Transfer to Design Validation System option helps to analyze the optimized model without having to set up the model again for material selection and boundary conditions.
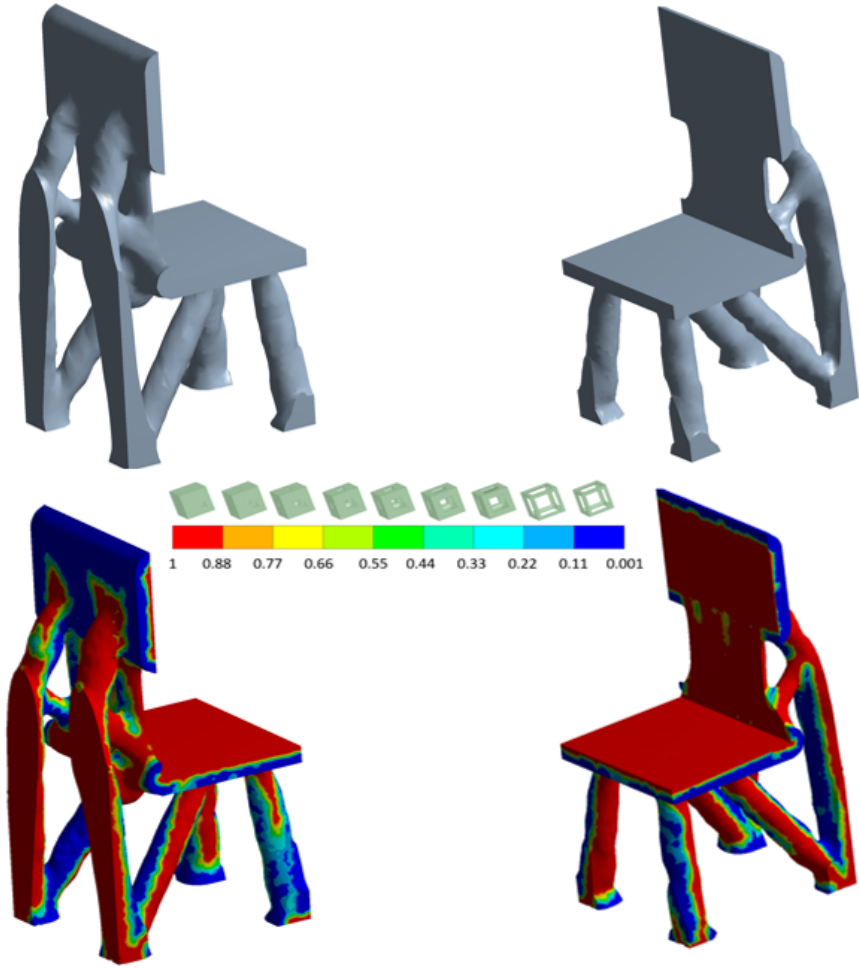
Figure 3: Optimization results. The top row shows the chair's organic shape obtained after Level set-based topology optimization. The bottom row shows Density contours for homogenization-based lattice topology optimization with cubic lattice with the corresponding legends shown.

19. Double click the model in the new Static Structural (Structural Analysis II) mesh the model similar to Structural Analysis-I and click Solve under the Home section in the top left corner. The results can be analyzed similar to steps 7-9.

20. To perform homogenization-based lattice optimization, go back to project schematic → Drag and drop Topology Optimization to Solution of Structural Analysis-II.

21. Update the project and select Setup in the Topology Optimization project.

22. 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 Lattice optimization → Lattice type: Cubic, Minimum Density: 0.001, Maximum Density: 1, Lattice cell size: 4 mm.

23. The homogenization procedure is done automatically without the need for the user to perform the analysis.

24. Select Objective → Response type → Compliance in the work sheet shown in the right.

25. For analyzing weighted multi-load case right, click on the worksheet → Add. Select the appropriate Step numbers corresponding to the loading cases in Start Step and End Step. The corresponding weights for optimization can be appended under the Weight column.

26. Select Response Constraint → Definition → Response → Mass → Percentage to Retain → Select the requisite value (Here a retention mass of 60 % is chosen) → Click Solve under the home section in the top.

27. To view the optimal nodal densities → Select Lattice density in the Solution section.

28. The final optimal Compliance can be seen by selecting Solution Information under Solution. Select Solution Output: Optimization Output to get the optimal compliance values or Optimization & Mass Response Convergence to analyze the convergence characteristics.

29. The results of the multi-scale optimization are shown in fig3.