Replication of results

This section describes the step wise procedure to do topology optimization of C-clip using Altair OptiStruct 14.0. The C-clip is fixed at left end and loads are applied in 'z' direction at inner side of right end. The magnitude of load is 100 N and applied at last five nodes of both top side and bottom side. The optimization problem is defined as minimizing the compliance along with a volume constraint, i.e., allowable volume is 50% of the total volume. The design domain and non-design domain are differentiated with orange and gray colors respectively.

- Importing the geometry file:
  Go to File → Import → Geometry → choose the geometry file (ROR-Geometry.stp) → Import. Rename it as 'Design'. Figure 1 shows the geometry after the import.

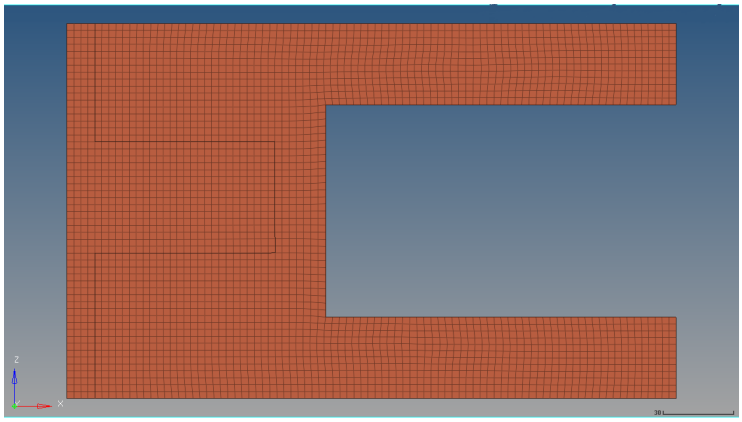

Figure 1: Geometry without nondesign domain and no loads

- Meshing the geometry:
  Click on 3D → tetramesh → choose the component → click on mesh → return.

- Creating component, material and property and assigning it to the geometry:
  Creating a material: Materials → give mat name = 'steel' → choose type = 'isotropic' → card image = 'MAT1' → then click on create → return.
  Creating property: Property → give prop name = 'Design' → type = '3D' → card image = ' PSOLID' → material = 'steel' → click on create → return. Repeat this procedure for creating 'Non-Design' property by giving prop name as Non-Design.
  Creating a component : Component → comp name = 'Non-design' → property → 'Non-Design' → create → return.
  Assigning elements to Non-design component: Mesh → Organize → Elements → To Component → choose elements from the components → dest component ='Non-design' → move → return.
  Assigning properties to Design component: Components → click Property → choose Design → OK; click Material → choose steel → OK.

- Creating and applying the loads:
  Creating the displacement BC's : Load Collecters → loadcolname = 'Fixed' → create → return. Repeat the same procedure with loadcolname = 'Loads' to create load.
  Applying the displacement BC's : Make sure that current select load is 'Fixed', other wise right click on it and choose 'Make Current'. To apply displacement constraints click on Analysis → constraints → nodes → select the nodes → tick on dof1 to dof6 → create → return.
  Applying the force BC's : Make the 'Loads' as the current collecter. Analysis → forces → select nodes → choose = 'constant components' → give X, Y, Z components of force → create → return. Figure 2 shows the geometry includes the non-design domain after applying the boundary conditions.

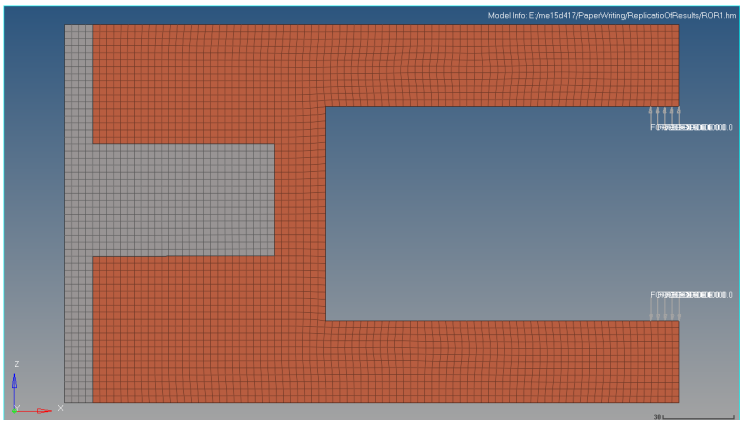

Figure 2: Geometry with nondesign domain and loads

- Creating loadsteps:
  Analysis → loadsteps → name = 'LS' → choose SPC = 'Fixed' and LOAD = 'Loads' → type = 'linear static' → create → return.

- Setting optimization paramets:
  Analysis → optimization → topology → create → desvar = 'TopOpt' → type = 'PSOLID' → props → Design → select → create → return.
  responses → response name = 'volfrac' → response type → choose volfrac → create → return. Repeat this to create another response, i.e, compliance.
  dconstraints → constraint = 'volcon' → tick on upper bound → give the volume fraction value as 0.5 → response = 'volfrac' → create → return.
  objective → choose min → response = 'compliance' → loadsteps → LS → create → return → return.

- Running the OptiStruct:
  Analysis → Optistruct → run options = 'optimization' → click on OptiStruct.

- Viewing results:
  Analysis → OptiStruct → HyperView. This opens HyperView software.

To view results, click Iteration to last number using the drag and drop bar, then go to Iso → current value = '0.6' → Apply. Figure 3 shows the optimal design obtained for the boundary conditions.

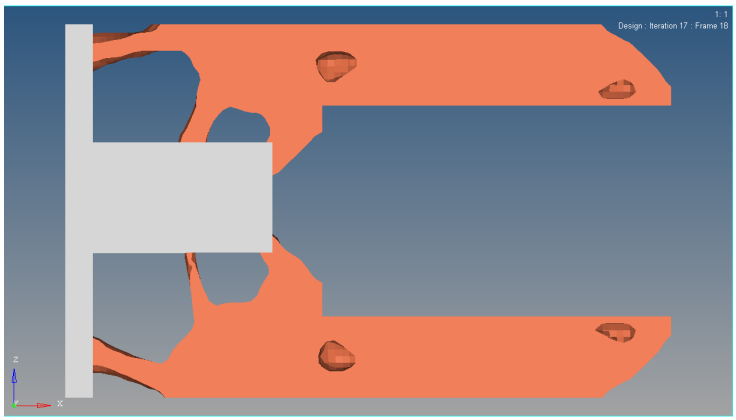

Figure 3: Optimal design obtained for density threshold value of 0.6

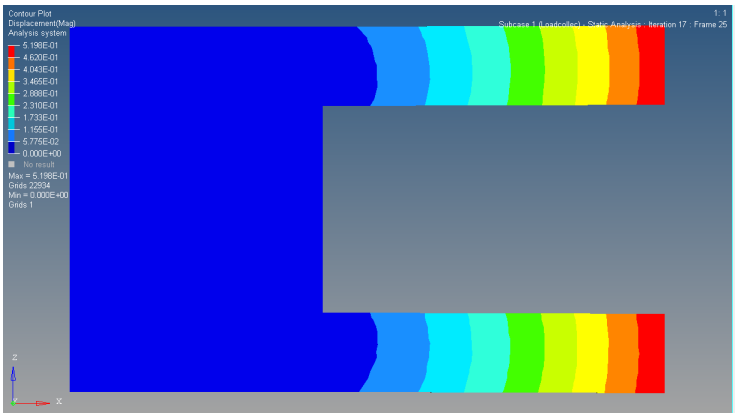

Figure 4: Displacement contours of the optimal design

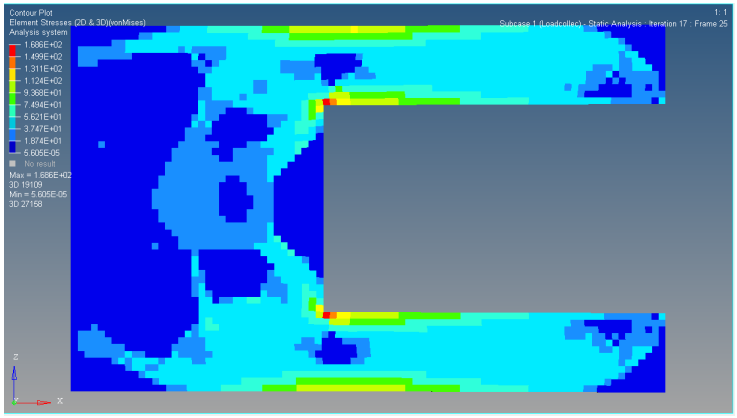

Figure 5: Stress contours of the optimal design