**ANSYS Fluent Meshing Watertight Geometry Workflow**

Workshop 9: High-Speed Train

The objective is to generate a CFD-ready volume mesh to study the external aerodynamics of a generic High-Speed Train geometry. The learning objectives are to create Local Refinement Regions, Share Topology, assign dead regions to avoid modeling, and add boundary layers with a specified first cell height.

For external aerodynamics simulations, like flow over an aircraft, a car, or a train, care must be taken to generate meshes for accurate predictions of aerodynamic forces like lift and drag. Fig. 1 shows the train geometry placed in a wind tunnel.

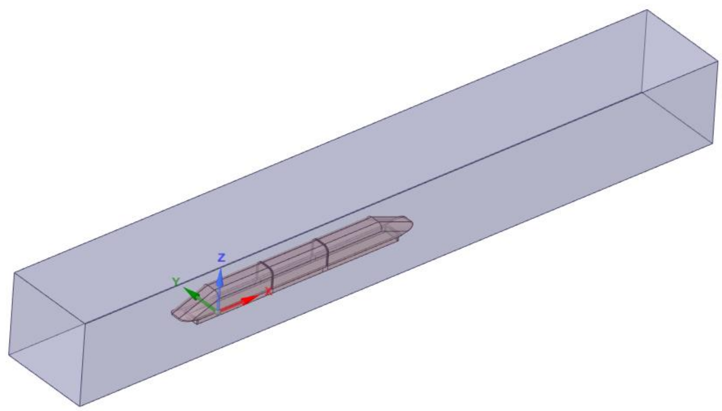

Fig 1: Imported geometry of Train in a wind tunnel

In the meshing, double precision is used along with 4 processors. In the local sizing, local refinement regions were created using the 'Box' method. This method allows the user to construct a box around the selected geometry to define specific mesh sizes. In this method, the limits need to be assigned in each direction as a ratio relative to the length of the selected geometry in that specific direction. Fig. 2 shows the refinement box created around the train walls.

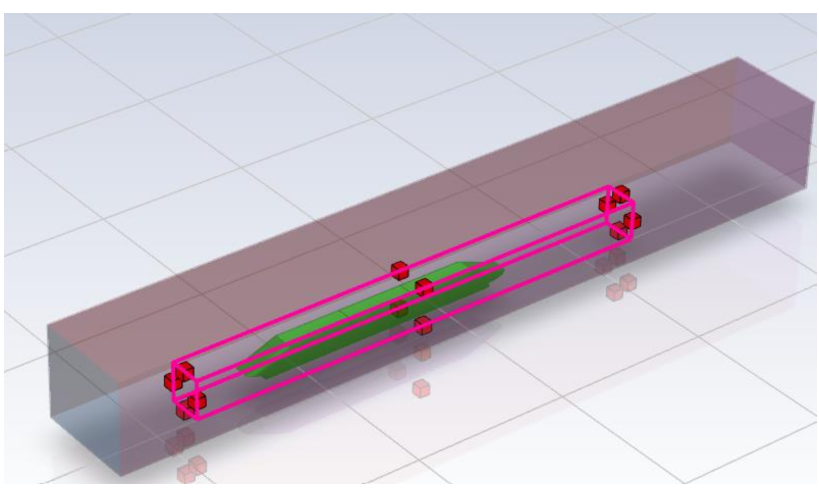

Fig 2: Local refinement region around the train walls

In the update boundaries task, Fluent automatically assigns the appropriate boundary conditions based on the label name. To avoid meshing inside the train geometry, the region has been assigned as a 'dead 'region type. Figure 3 shows the surface mesh generated in the enclosure and the refined region in the refinement box. Figure 4 shows the surface mesh generated on the surface of train.

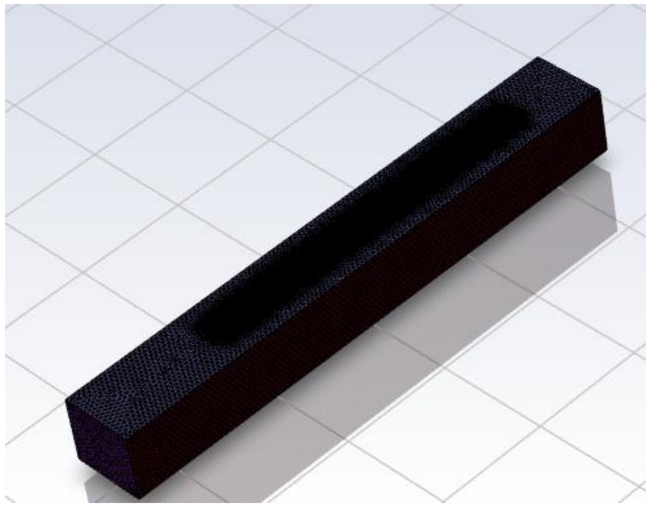

Fig 3: Surface mesh

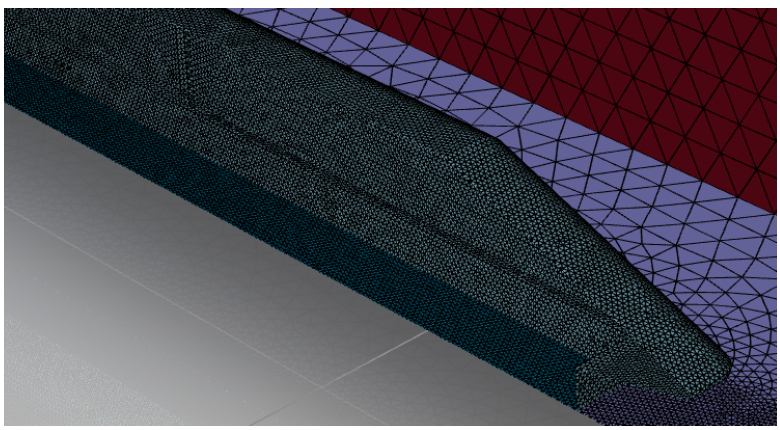

Fig 4: Surface mesh generated on the train's surface

After the initial surface mesh generation as shown in Fig. 3, 'Apply Share Topology' is automatically added in the task section. The fluent recognizes that it has not been created during the CAD generation stage and hence it adds this task to the workflow. In Fig. 5, the interfaces between the train and enclosure surface are marked for sharing.

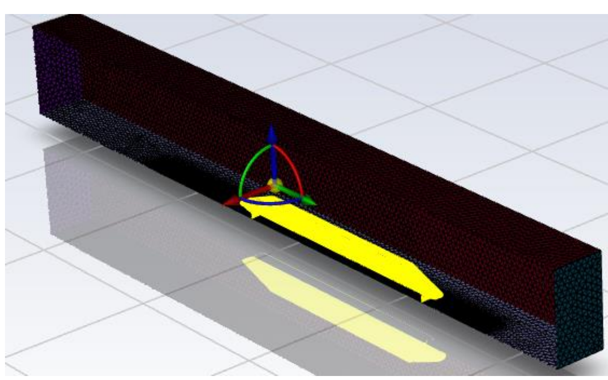

Fig 5: Highlighted interface for the 'Share Topology'

For the boundary layer generation, the last-ratio method has been used. The last-ratio method provides the option to control the first height to control the y+ value of the mesh. The transition ratio allows for control of cell size transition between the core mesh and boundary layer level mesh. Figure 6 shows the boundary layer generated on the boundaries of train.

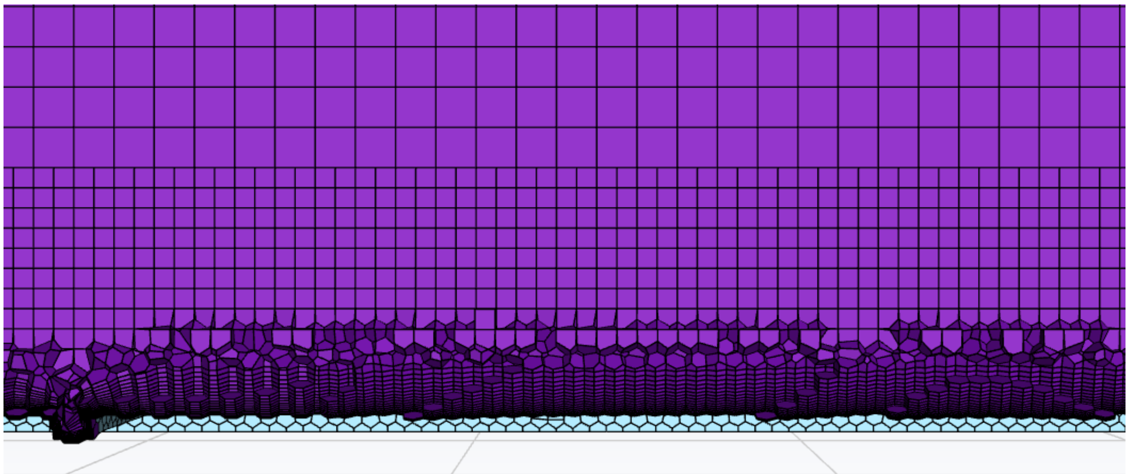

Fig 6: Boundary layer generated on the train walls

The first height is set as 25 mm. The transition ratio was kept at the default value of 0.272. For the start, the number of boundary layers was kept at 15 to ensure proper capturing of the boundary layer flow behavior, but it can be increased if more precision is required in external aerodynamics.

The poly hex-core method is used to fill the computational domain, which is connected to the boundary layer mesh with polyhedral cells. Figure 7 shows the volume mesh generated in the enclosure.

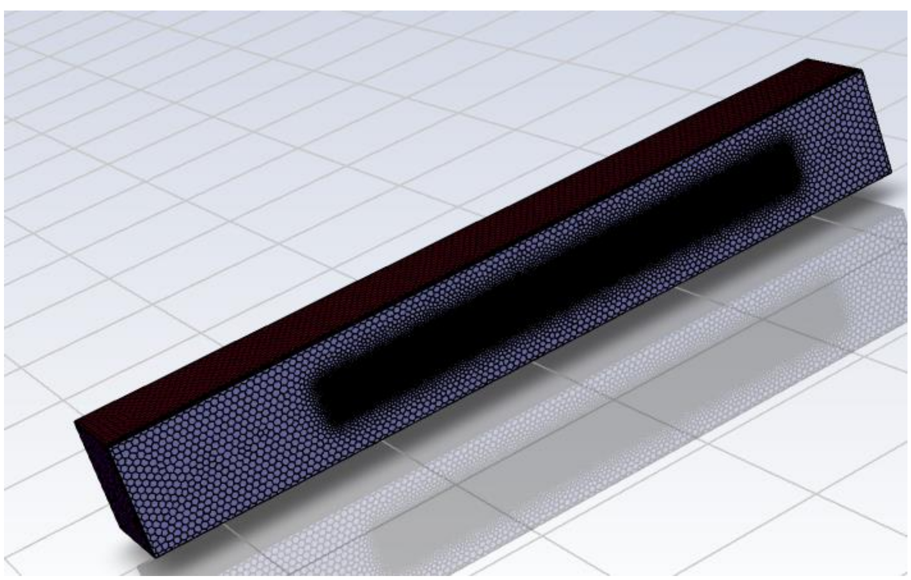

Fig 7: Poly hexcore mesh in the computational domain

The initial mesh quality was 0.07, which later on was improved by adding the task 'Improve Volume mesh' till 0.15 with minimal extra mesh count generation. The overall mesh count is 2205571.

Figure 8 on the next page shows the cut section of the volume mesh in the plane of the train. There is no mesh generated in the train region as the region is declared ‘dead’ while meshing. The region present in the refinement box has been refined to capture the wake regions behind the train.

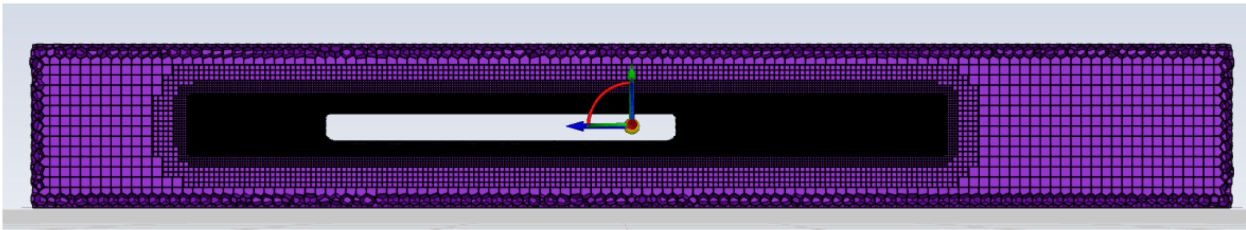

Fig 8: A cut-section of mesh showing refinement region and poly hex-core mesh in the domain