

ANSYS Fluent Meshing Watertight Geometry Workflow

Workshop 8: Mixing Tank

The objective is to generate a CFD-ready volume mesh for the mixing tank. The learning objectives are to apply share topology, region-based meshing, and extruding the boundary surfaces of the volume mesh.

Fig. 1 shows a constant volume mixing tank with a tangential inlet, outlet, and two impellers. These impellers are mounted on a central shaft and are used to mix the content inside the tank. For the CFD simulation, only the fluid flow region of the mixing tank is considered.

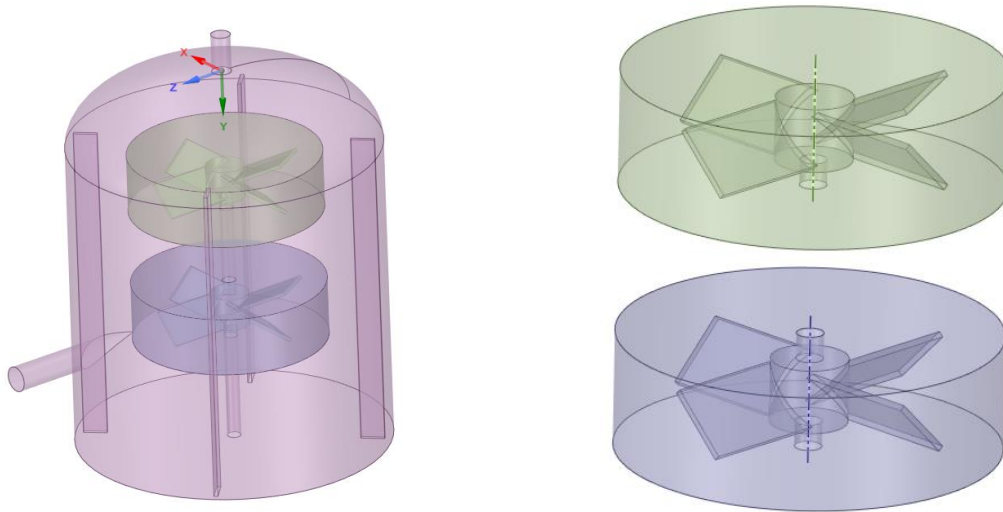


Fig 1: CAD geometry of the Mixing tank and 2 impellers in the tank

The rotation of the impellers is modeled using the Multiple Reference Frame (MRF) model. This is a steady-state approximation in which individual cell zones can be assigned different rotational or translational speeds. Hence, separate fluid regions are required around the rotating parts to differentiate the moving zones from stationary zones.

In the sectional view, shown in Fig. 2, there are three zones. Two zones are around the rotating impellers and the third zone is the remaining computational domain.

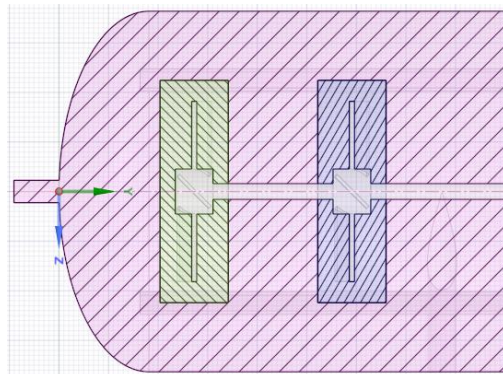


Fig 2: Sectional view of the tank

All the fluid-fluid interfaces should be changed to internal, as the interfaces are not physical, and fluid should be allowed to move across them.

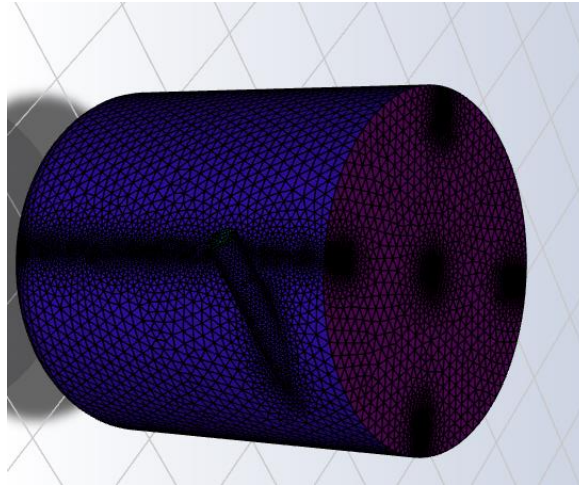


Fig 3: Generated surface mesh

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. 4, the interfaces between the moving and stationary zones are marked for sharing.

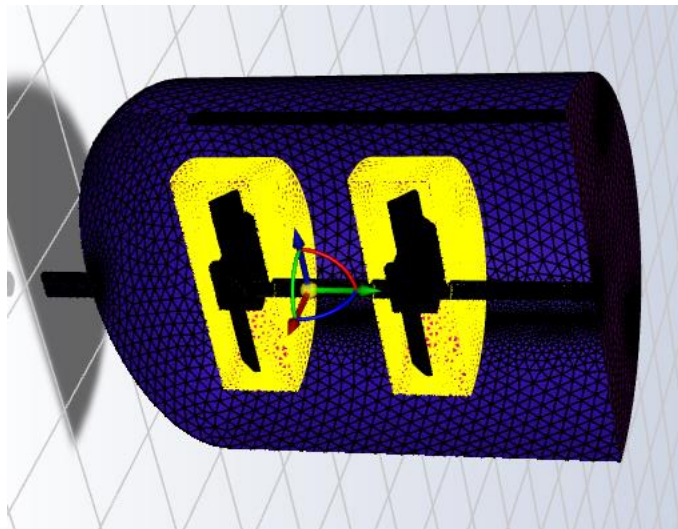


Fig 4: Interfaces for Share Topology operation

Poly hex-core is used for volume meshing, which provides mesh with a minimum orthogonal quality of 0.2 and a mesh count of 1.19 million cells. It was preferred over the Polyhedra 'fill with method' as the quality of mesh reduces to 0.07 in the latter one. Figure 5 shows the overall volume mesh generated in the mixing tank. Figure 6 and Fig. 7 show the sectional view of mesh and a close-up of mesh generated in the regions close to the impeller, respectively.

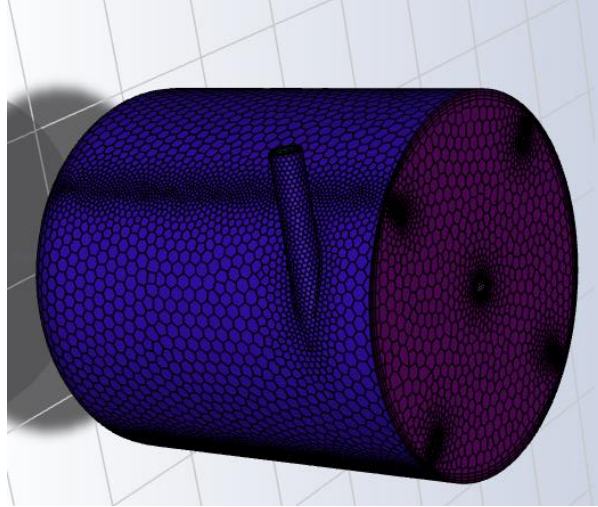


Fig 5: Poly hexcore mesh for the domain

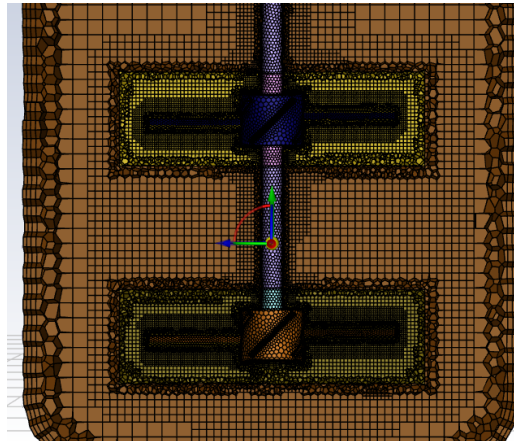


Fig 6: Sectional-cut view of the mesh

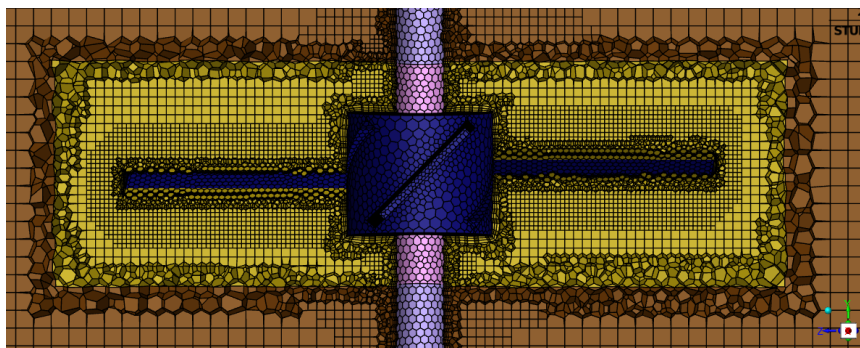


Fig. 7: A close-up view of the mesh near impeller

The sizing method is selected as 'Region-based Meshing' to control the meshing parameters in different regions. It is different than the BOI meshing used in Fluent, as in BOI mesh size is set for both the surfaces and interior whereas, in Region-based meshing the mesh size is set for only the interior. If a finer mesh is required at the face of the impeller wall, it can be provided using the face size control.

To allow for fully-developed flow in the inlet and reduce the backflow in the outlet of the mixing tank, the inlet and outlet were extruded using the 'Extrude Volume Mesh' task after volume meshing. Figure 8 shows the extruded inlet and outlet. During this operation, the inlet and outlet automatically shift to the new location.

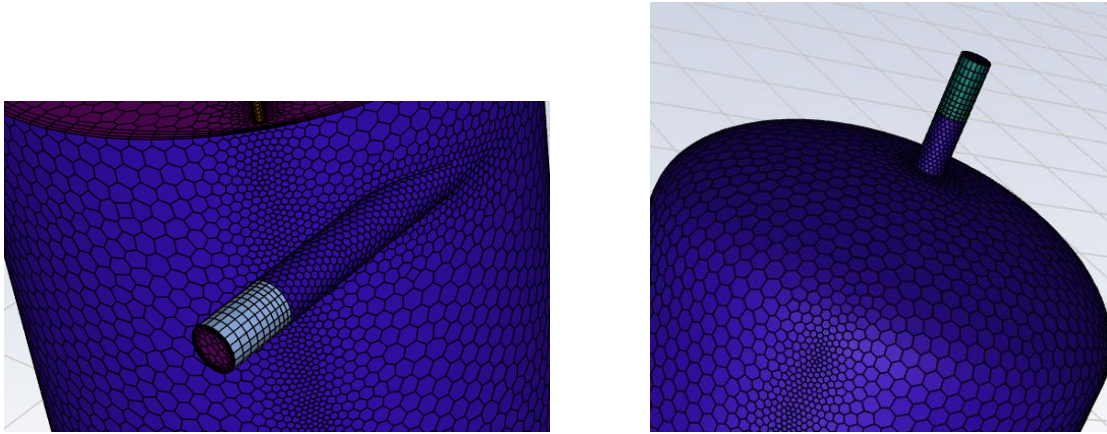


Fig. 8: Extruded inlet and outlet volume mesh

One thing to note is that, being a conformal mesh this mesh can be used only for MRF simulations and the setting needs to be changed in Fluent Solver to use the sliding mesh approach.