ANSYS Fluent Meshing Watertight Geometry Workflow

Workshop 10: Turbine Vane geometry

The objective is to generate a CFD-ready mesh for a generic Turbine Blade to study its conjugate heat transfer characteristics. The learning objectives are – curvature local size control to accurately resolve the curvature of cooling passages, employing manual rotational periodic boundaries, and setting up boundary layer mesh in both fluid and solid regions to facilitate an accurate prediction of temperature gradients.

The basic problems during the meshing of the Turbine blade are –

- The complex turbine blade geometry may lead to problems during the CAD creation phase.
- Problems can be related to fluid extraction during the meshing phase.
- Share topology can create problems during the CAD and meshing phase.

The following things can be addressed using the fluid domain extract feature and the workflow of ANSYS Fluent watertight geometry. Figure 1 shows the simplified turbine blade geometry along with the hub. The cooling channels inside the turbine blade is represented by 10 holes in the hub and geometry as shown in the Fig 1.

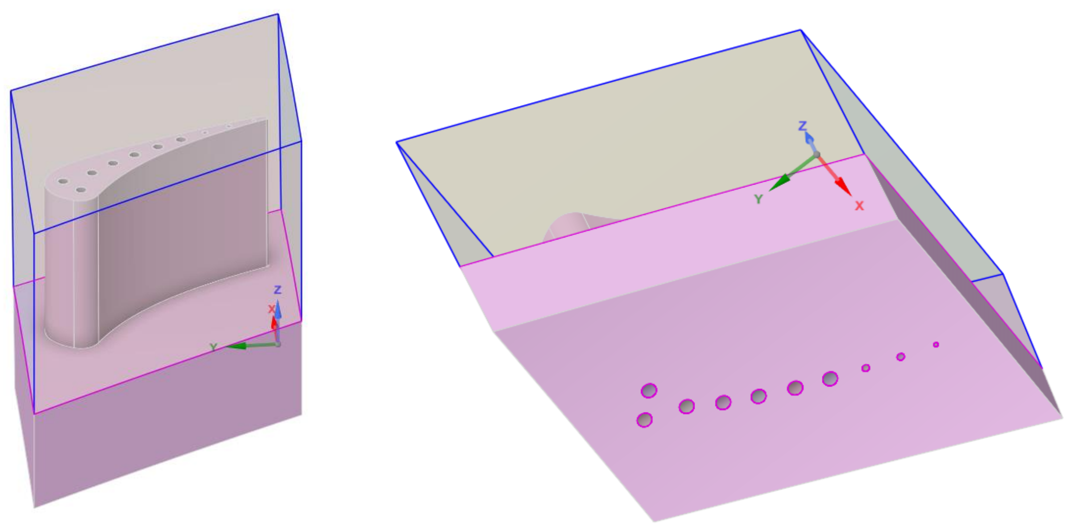

Fig 1: CAD geometry of the Turbine blade attached to the hub

The blade is enclosed by surfaces that make up the surfaces of the fluid domain. Figure 2 shows the surface mesh generated on the walls of the domain. Figure 3 shows the surface mesh generated the turbine blade and cooling channel walls.

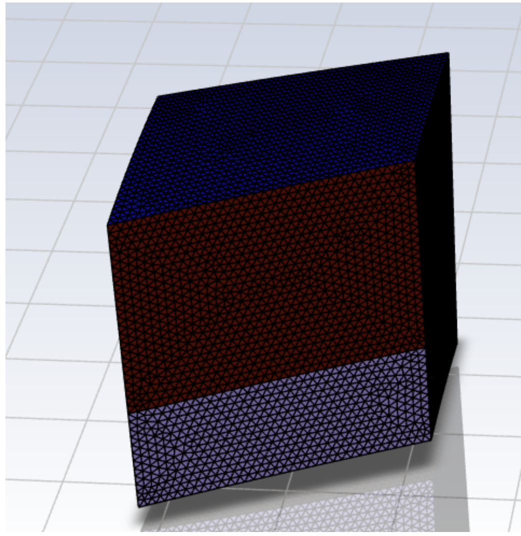

Fig 2: Surface mesh generated onto the walls of the domain

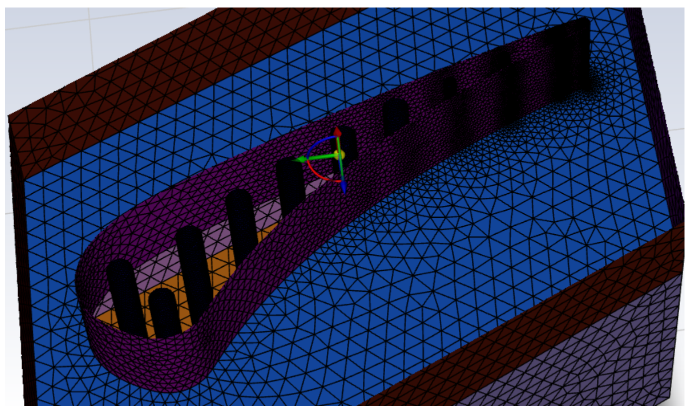

Fig 3: Surface mesh on the turbine blade and cooling channel walls

Boundary layers were added in the fluid regions. For CHT analysis boundary layers can be added for better temperature gradient prediction. A smooth transition mesh was used for growing boundary layers. This method allows us to control the transition ratio and growth rate of the inflation layer.

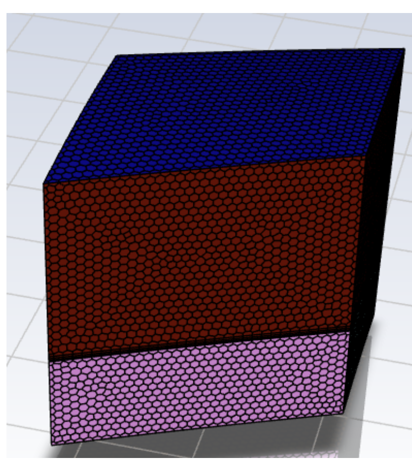

Fig 4: Conformal volume mesh in fluid and solid regions

The transition ratio is the ratio between the cell area of the last layer of inflation and the first cell area out of the inflation, whereas the growth rate represents the increase in element edge length with each succeeding layer of elements.

For the volume mesh generation, Poly hex-core volume ‘fill with method’ is used. The overall mesh quality was 0.2, which was above the recommended value with an overall mesh count of 1.25 million cells. Figure 4 shows the conformal poly hex-core volume mesh generated in both the fluid and solid regions. Figure 5 and Fig. 6 shows the cut-section of overall mesh, and mesh generated in the turbine blade region along with cooling channels.

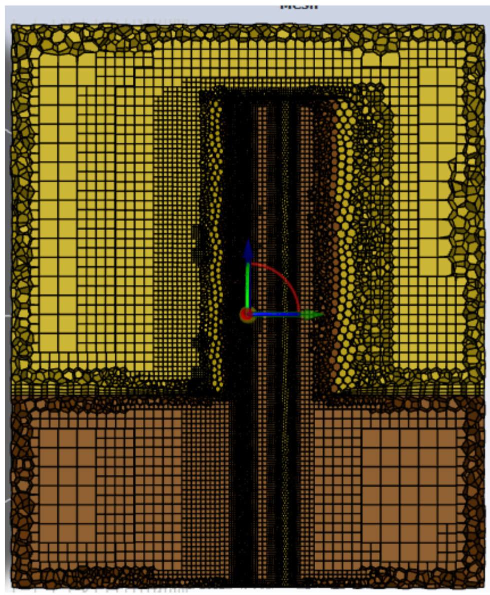

Fig 5: A cut-section of the mesh showing both fluid and solid regions

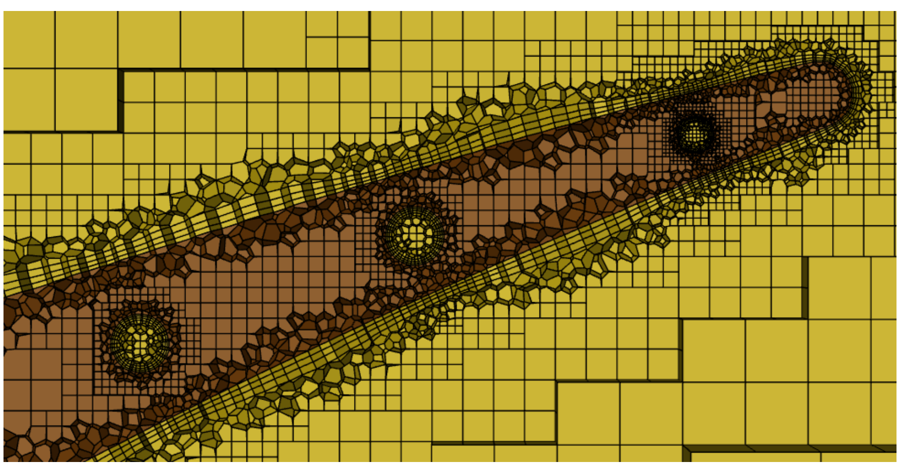

Fig. 6: Mesh generated in the blade region and in the cooling channels