ANSYS Fluent Meshing Watertight Geometry Workflow

Workshop 1: Axial Fan

ANSYS SpaceClaim operations:

The aim of this workshop is to mesh an axial fan, which is a symmetrical object, and thus explore the use of periodic boundaries in ANSYS water tight geometry workflow.

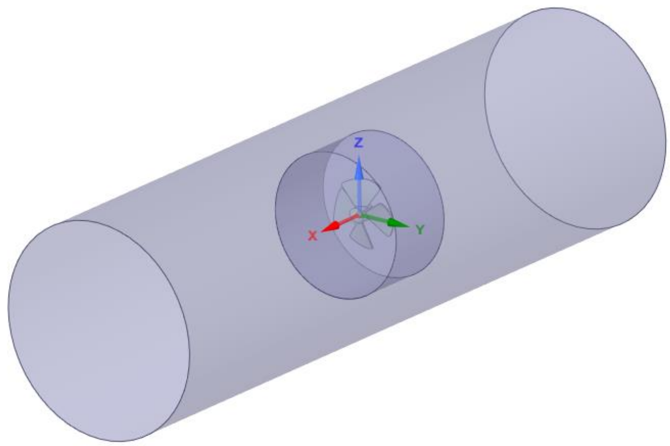

Fig 1: Available CAD geometry of the Axial Fan

Fig 1 shows the given CAD geometry. It has three parts, the axial fan, a small enclosure for the axial fan, which is used for refining the surrounding regions of fan, and a large enclosure for modeling the computational domain.

The axial fan in this domain is treated as void region and there no mesh is generated for it.

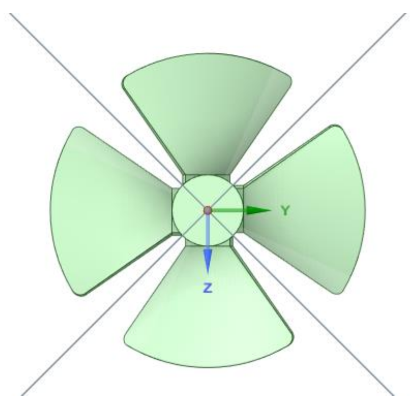

Fig 2: Creation of a cut section of Axial Fan

Fig 2 shows the process of setting up the computational domain for setting up the periodic boundary. The fan is cut into 4 parts, out of which only one part is used to generate the computational domain as shown in the fig. 3.

For setting the periodic boundary the surface area for both the sides should be same.

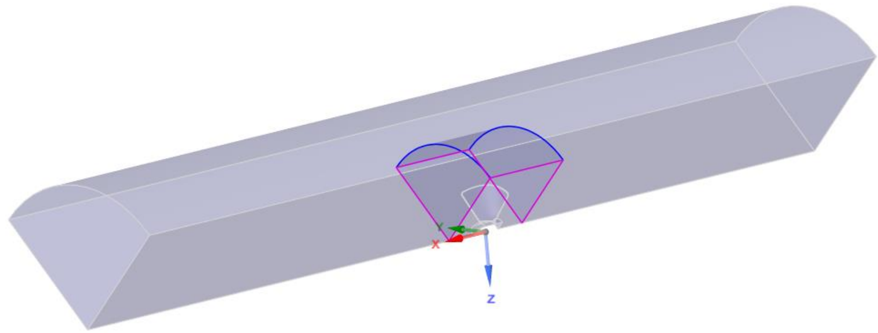

Fig 3: The computational domain after the section cut

ANSYS Meshing operations:

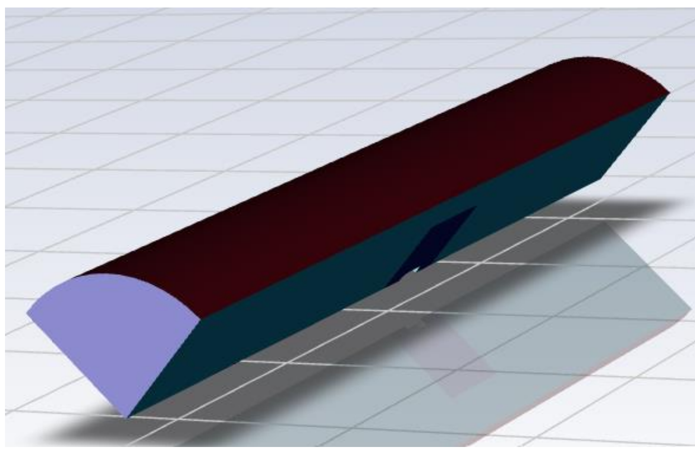

Fig 4: Imported geometry in the ANSYS Meshing

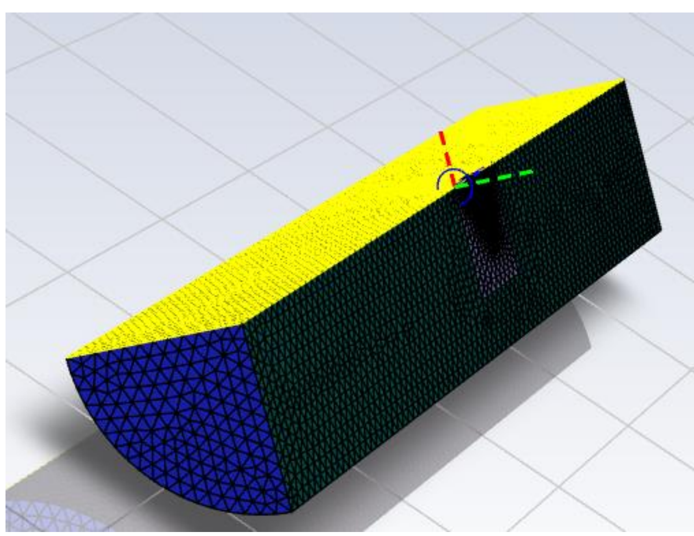

Fig 5: The surface mesh

The geometry is imported in ANSYS Meshing, shown in fig. 4 and the surface mesh is generated keeping the minimum possible size of mesh required to refine the axial fan geometry.

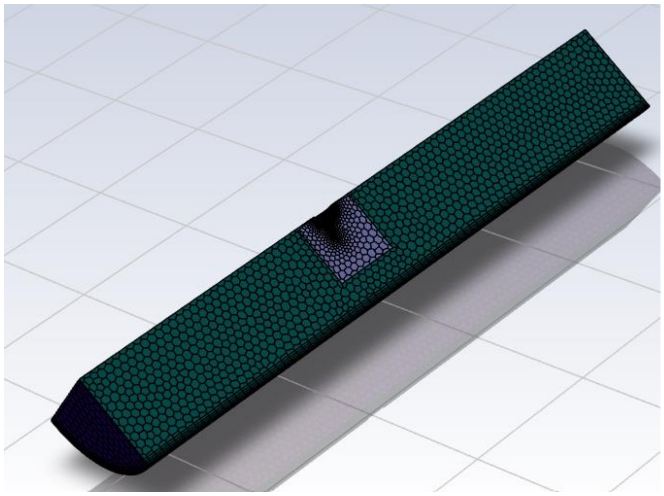

Fig 6: The volume mesh

The other operations before generating the volume mesh, are setting up the periodic boundaries, fluid regions, boundary layers.

The initial height of the boundary layer is decided based on the minimum height required to refine the boundary layers on the wall of the pipe. A total of 5 layers are used in boundary layers.

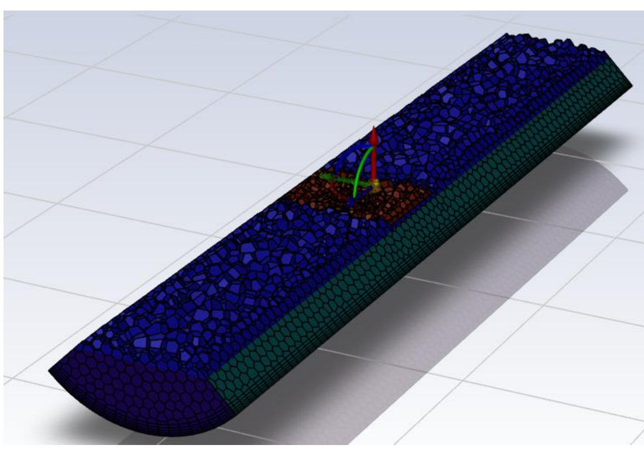

Fig 7: Cut-section showing the volume mesh

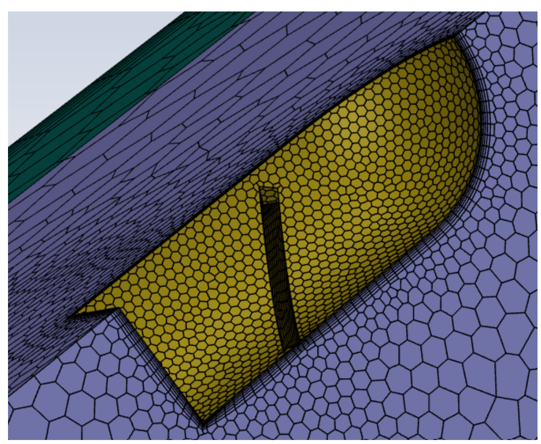

Fig 8: The zoomed section showing the axial fan

Fig 7 and fig 8 shows the cut – section and the zoomed section near the axial fan geometry.

For the meshing purpose, all four types of volume meshing methods available in the workflow is used. The best method is selected based on the orthogonal quality and the total mesh count.

As it can be seen from Table 1, polyhedra have the best orthogonal quality and the least mesh count. The primary reason for polyhedra having the least mesh count is that polyhedra cell is made by combining several tetrahedral cells, and hence it has more number of faces as well.

As a result, it has a better flow variables gradient for flow coming from any direction, and therefore very low artificial diffusion gets introduced in the solution.

Type of volume mesh	Min Orthogonal Quality	Mesh count
Tetrahedral	0.1	258574
Polyhedra	0.21	77866
Hexcore	0.1	319025
Poly-Hexcore	0.2	103163

Table 1: Orthogonal quality and mesh count in all the four meshing methods

As a final process, the overall mesh is transformed to include all the four sections of the geometry. This is seen in the fig. 9. During this transformation, all the fluid-fluid boundaries have been changed to internal.

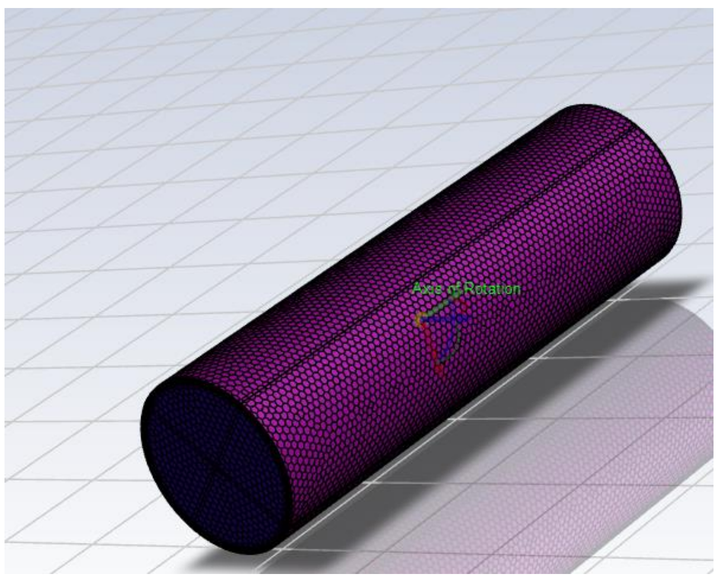

Fig 9: The complete computational domain