ANSYS Fluent Meshing Watertight Geometry Workflow

Workshop 7: Catalytic Converter

The objective is to generate a CFD-ready mesh for the Catalytic Converter. It is an exhaust emission control device generally used with IC engines. The objective is to learn to apply the region-based sizing method in the "Volume fill with" option, to rename zones after generating the volume mesh, and to extend boundary zones after meshing is done to prevent backflow.

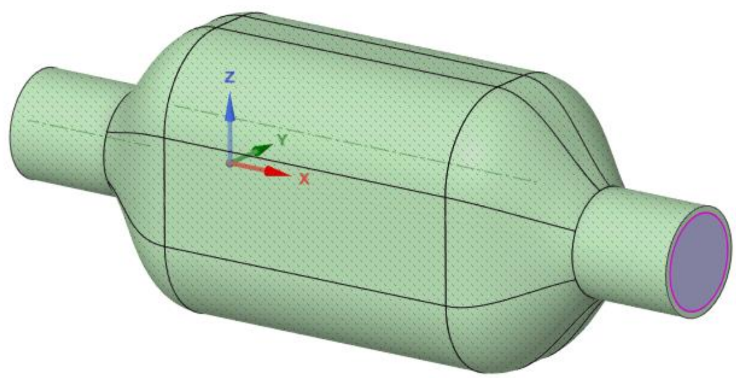

Fig 1: CAD geometry of the Catalytic Converter

Fig.1 shows the available CAD geometry of the Catalytic Converter. There are three regions in geometry. Starting from the left is the inlet, then the porous region, and then the outlet. Fig. 2 shows all the regions in the imported geometry in the ANSYS meshing. The porous region is highlighted in green.

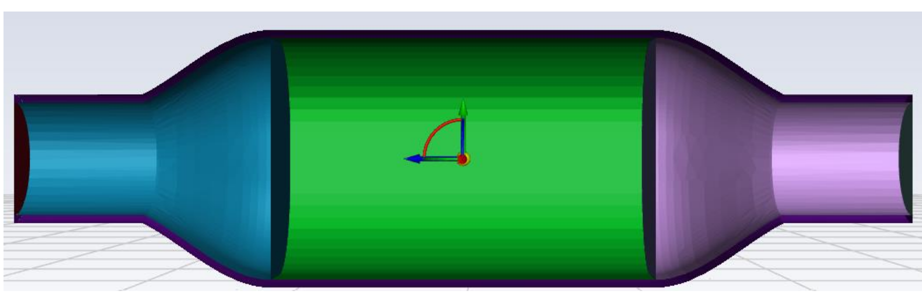

Fig 2: Imported geometry in ANSYS meshing showing all regions

To generate the surface mesh, the following input parameters were given:

- Minimum size = 0.34 mm
- Maximum size = 4 mm (approximately 10% of the inlet diameter)
- Cells per gap = 2
- Separate Out Boundary Zones by Angle = yes (40 degrees)

If there is no named selection for solid geometries defined at the CAD level, then the ANSYS meshing treats the whole body as a single entity. Hence if needed at later stages to perform different actions on different surfaces, which is in this case to extrude the solid and fluid regions outlet boundary, then "Separate Out

Boundary zones by angle" needs to be selected "yes". This command separates the solid boundary zones, and later some actions can be performed on them separately.

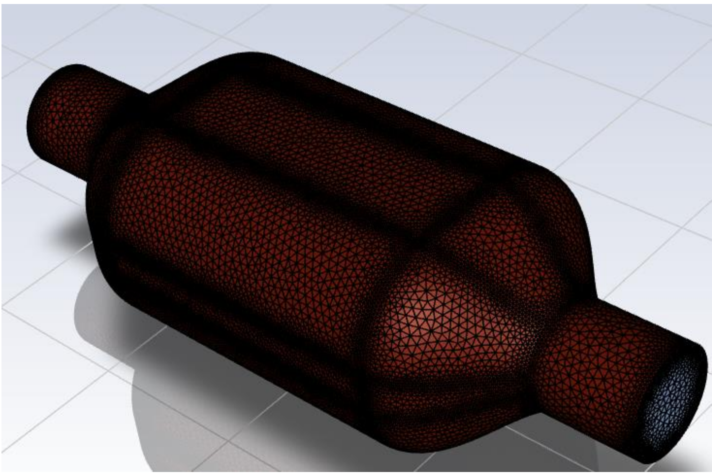

Fig 3: Generated surface mesh

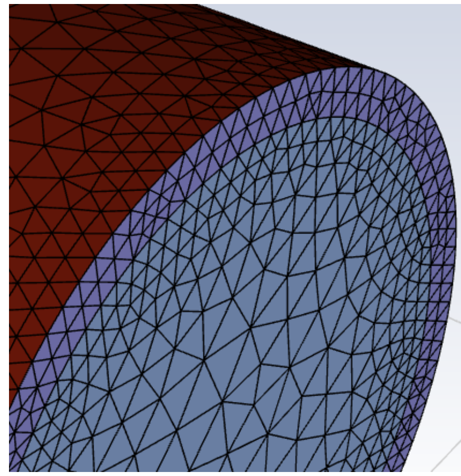

Fig 4: Minimum 2 cells per gap

Fig. 3 shows the generated surface mesh based on the above selected parameters. Fig. 4 shows the minimum 2 cells in the thickness of the solid. The number of cells can be increased based on the required accuracy for the measured variable.

In the "Describe Geometry" task, the input to change all fluid-fluid boundary types from "wall" to "internal" is selected "yes".

Boundary layers were added in both fluid and solid regions. A smooth transition mesh were used for growing boundary layers. This method allows us to control the transition ratio and growth rate of the inflation layer. 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.

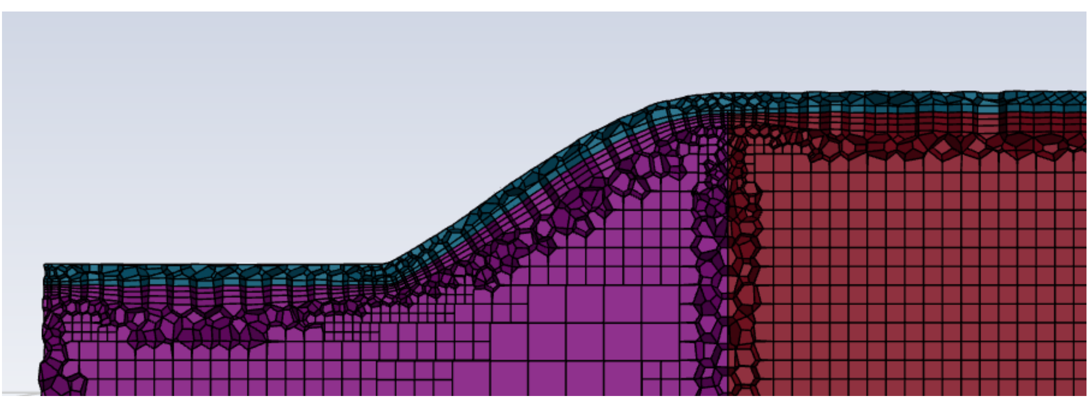

Fig 5: Boundary layers in the final volume mesh

In fluid regions, 3 boundary layers were used, and in solid 1 boundary layer was developed on the solid-fluid interfaces to avoid sharp jumps in the temperature values during conjugate heat transfer analysis. Fig. 5 shows the boundary layers present in the poly hexcore volume mesh.

Since the flow in the catalytic converter is wall-bounded and directional. Hence it is physically sensible to use poly hexcore mesh because the hex cells of the mesh will be aligned with the flow direction.

During the volume mesh method selection, the “Merge Back the Separated Boundary Zone” was selected as “no” to facilitate the extension of the solid boundary along with the fluid boundary after the generation of the volume mesh. The initial volume mesh was refined by changing the sizing method from “Global” to “Region-based sizing”. Fig. 6 shows the cut-section of the final volume mesh. The porous region has been refined to accurately predict the variable gradient in those regions.

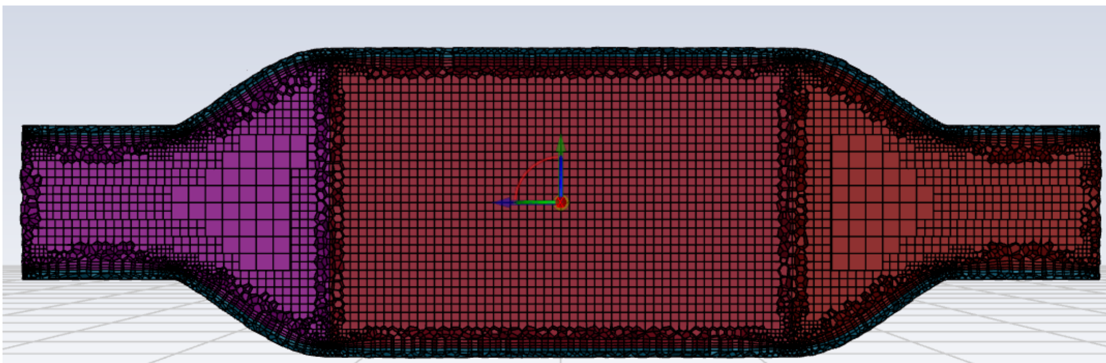

Fig 6: Cut-section of the final volume mesh

The overall mesh quality was 0.2, which was above the recommended mesh quality of 0.1. The overall cell count was 353014.

The zone name can be changed after generating the volume mesh. This can be done by inserting a new task in the “Generate the volume mesh” as “Manage zones”.

To prevent backflow during simulation, sometimes the outlet boundary needs to be extended. This can be done by adding “Extrude mesh task” in the workflow. Fig. 7 compares the two volume mesh, before and after the outlet boundary has been extruded.

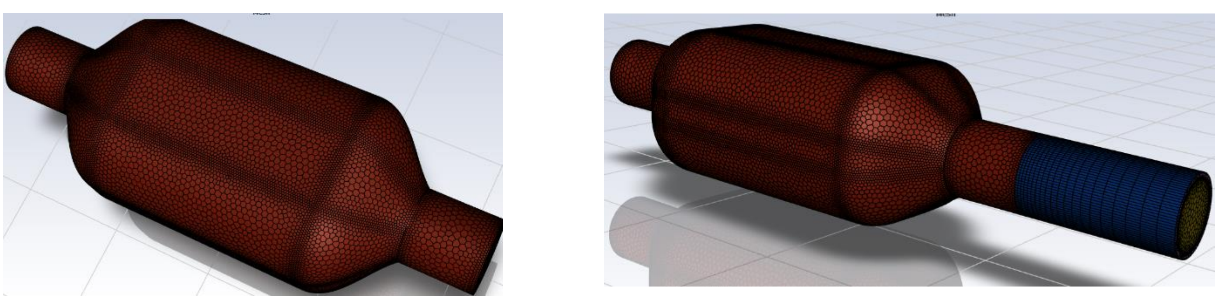

Fig 7: Comparison of the before and after extrude zone option

For extruding the zone, following inputs need to be provided:

- Total height of extrusion
- Total number of layers

- Growth rate

All the inputs have the usual meaning. The growth rate simply is the ratio of the edge of the next cell to that of the previous cell, whereas the total number of layers defines how many layers need to be created within the total height assigned. Fig. 8 shows the cut section of the extruded mesh.

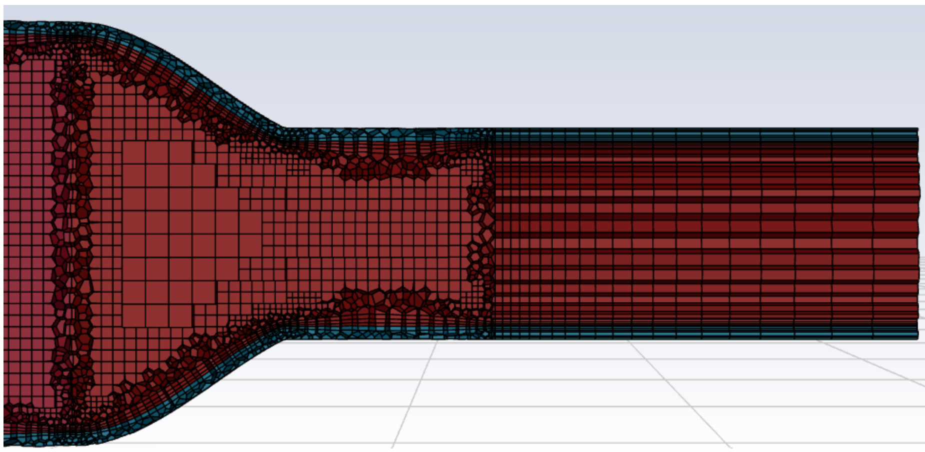

Fig 8: Cut section of the extruded zone