

Project – 5 Parametric study of the flow in Heat Exchanger

The objective is to perform a parametric study of the flow in the heat exchanger using the Python scripting in ANSYS PyFluent Core and Parametric. The learning objectives are –

Develop the Python script for ANSYS Watertight meshing workflow with the following steps –

- Import the geometry
- Generate the surface mesh
- Define geometry
- Update regions
- Add boundary layer – (by smooth transition method)
- Generate the volume mesh

For the Fluent solver –

- Viscous model selection
- Adding the boundary conditions
- Adding different fluid in different fluid regions
- Hybrid Initialization
- Solving the case file for performing a Parametric study using PyFluent Parametric
- Generate the report for the parametric study

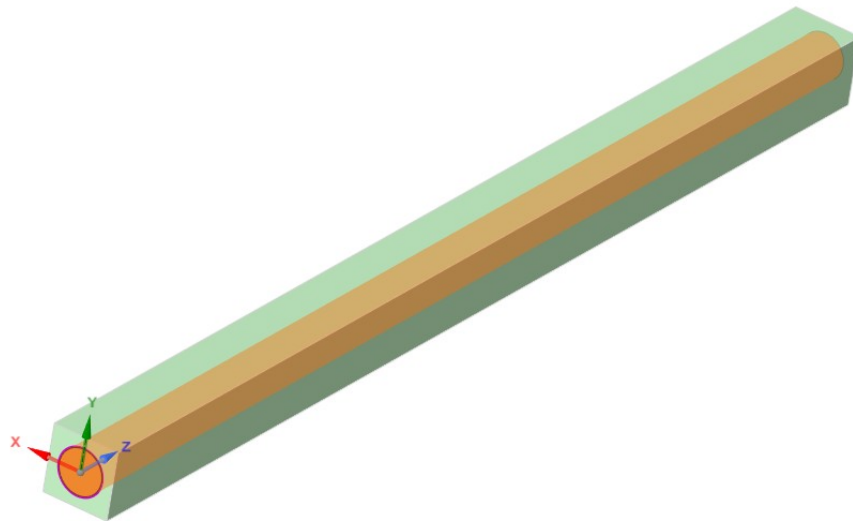


Fig: 1 CAD model for the simulation

The cad model above shows the computational domain of a cross-flow heat exchanger. The cross-flow heat exchanger consists of two fluid region, one cylindrical region having water at higher temperatures and the other rectangular fluid region containing air at lower temperatures. The two fluid regions are separated by 1 mm thick aluminum pipe.

For the parametric study, the temperature of the water is varied from 500K to 600 with 25 K increase at each step. The effect of the temperature increased is studied on the final temperature of the air and is measured at the air-outlet. The results were tabulated and then presented in the report.

Meshing -

- Orthogonal quality – 0.2
- Mesh count - 362997

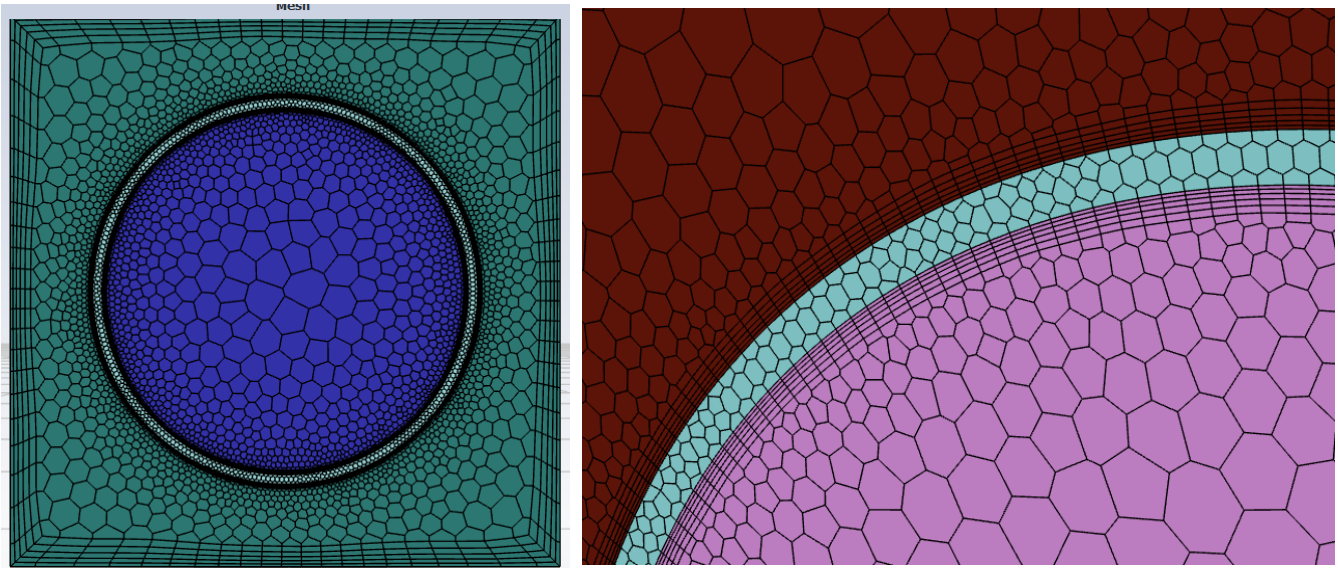


Fig:2 Conformal mesh at the cross-section of the pipe between all the three regions

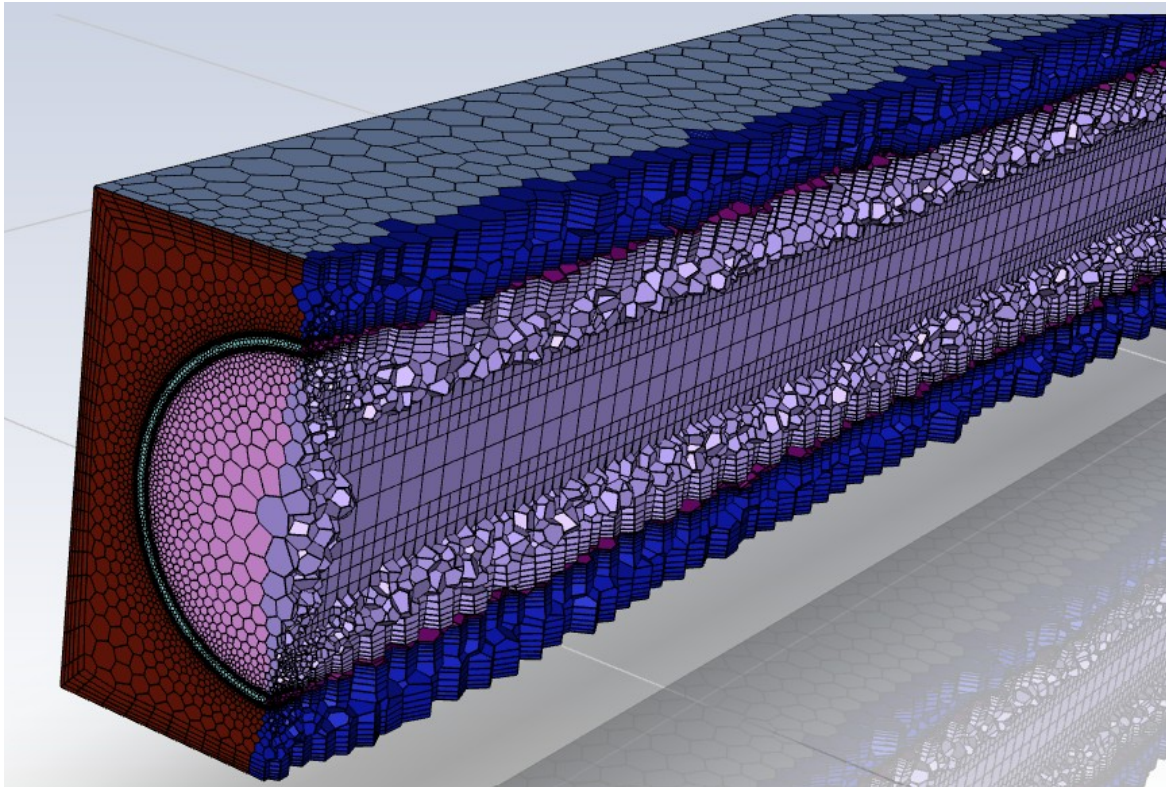


Fig 3 - Cut-section of the Poly-hexcore volume mesh generated inside the domain