

Project – 1

Semi-circular Obstacle in a channel

The objective is to simulate the flow over a semicircular obstacle using the python scripting in ANSYS PyFluent. The learning objectives are –

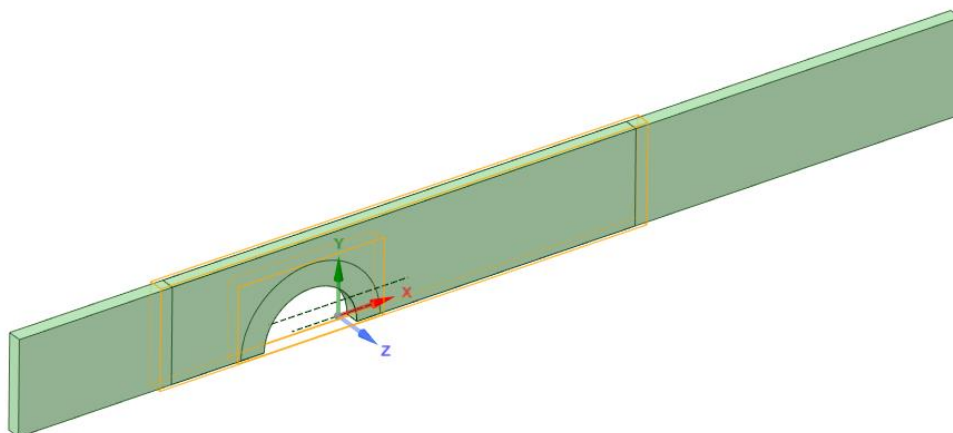
Develop the python script for ANSYS Watertight meshing workflow with following steps –

- Import the geometry
- Local size refinement (BOIs)
- Generate the surface mesh
- Define geometry
- Update regions
- Add boundary layer – (smooth transition method)
- Generate the volume mesh – (keeping the default size)

For the Fluent solver –

- Viscous model selection
- Adding materials (other than default)
- Adding the boundary conditions
- Initialization
- Solving the problem
- Plotting the contours using PyFluent Core
- Plotting the pressure and wall shear stress along a surface

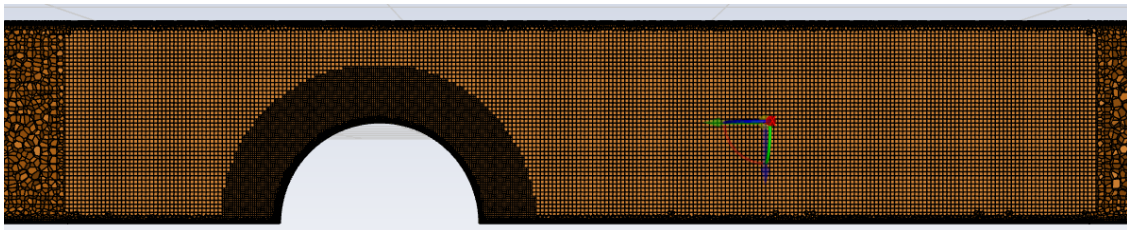
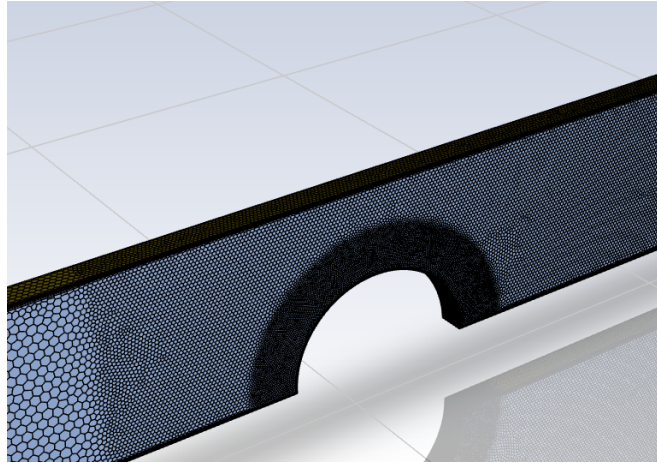
CAD model -

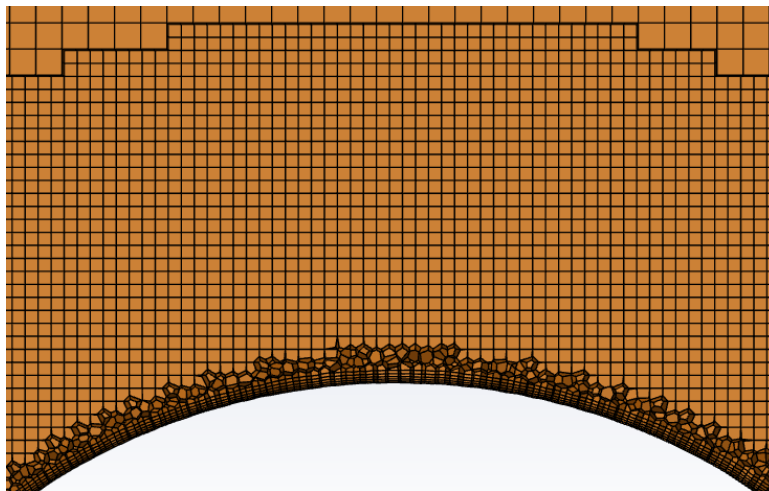
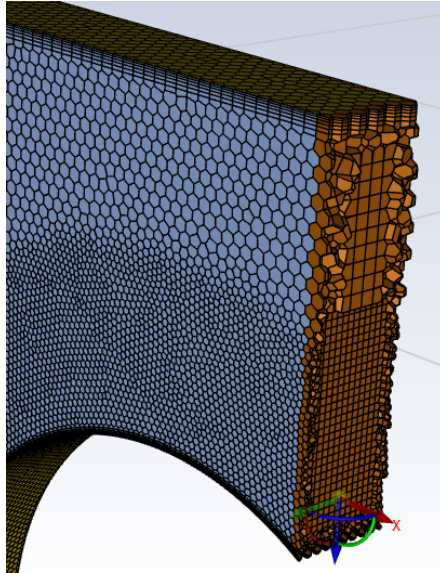


Geometric dimensions –

- Semi-circle radius – 50 mm
- Cross-section area – 100 mm * 15 mm
- Up-stream (semi-circle) length – 250 mm
- Downstream (semi-circle) length – 800 mm

Meshing Images -





Results –

- Fluid = water-liquid
- Inlet = 10 m/s flow velocity
- Viscous model = k-omega (SST)

