

Project – 4 Flow over a Circular Cylinder

The objective is to simulate the flow over a circular cylinder at $Re = 500$ using the Python scripting in ANSYS PyFluent. The learning objectives are –

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

- Import the geometry
- Local size refinement (BOIs)
- Generate the surface mesh
- Define geometry
- Update regions
- Add boundary layer – (by last ratio method)
- Generate the volume mesh – (change the mesh size)
- Extrude the mesh – (develop the script to extrude the mesh)

For the Fluent solver –

- Viscous model selection
- Adding the boundary conditions
- Transient flow simulation
- Hybrid Initialization
- Changing the relaxation factor for Pressure and Momentum terms
- Changing the discretization scheme for Momentum equations
- Solving the problem
- Plotting the contours using PyFluent Visualization
- Plotting the pressure and wall shear stress along a surface

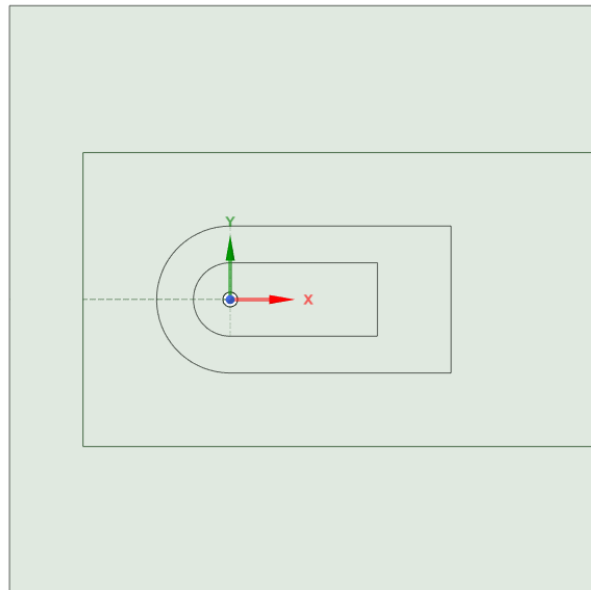
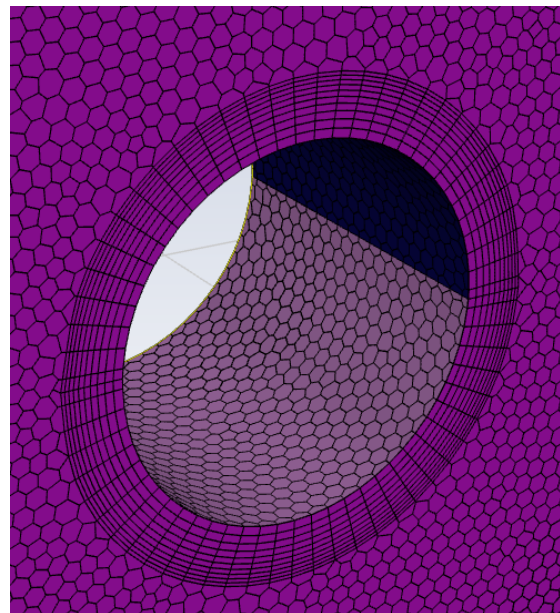
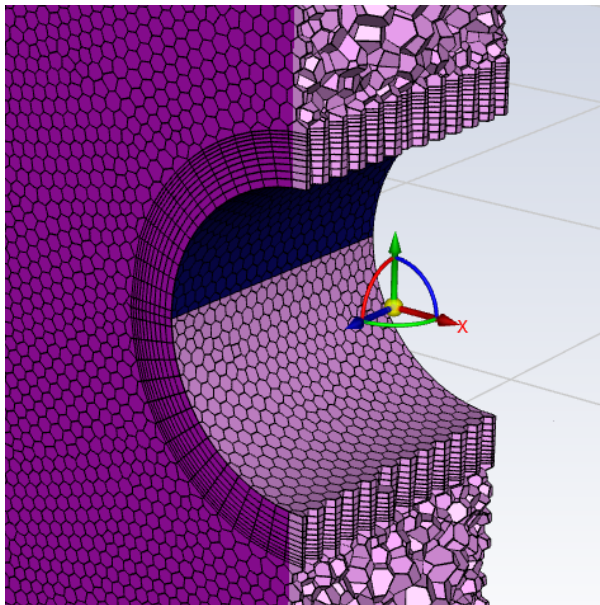
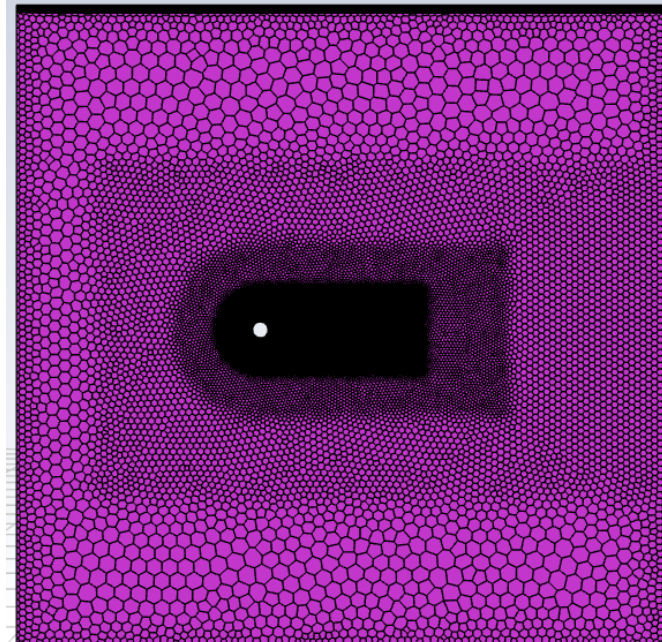


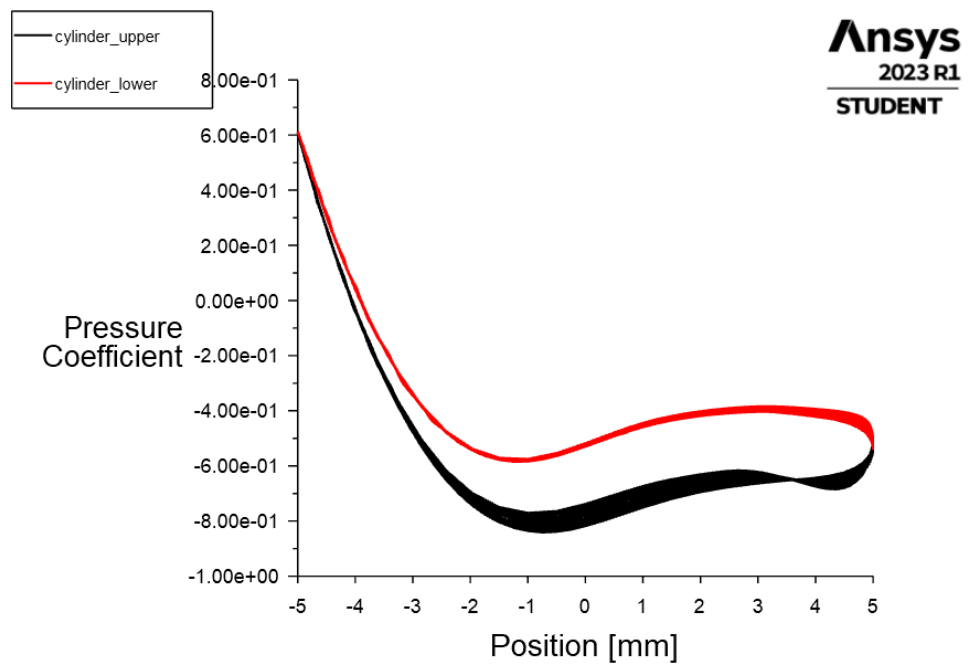
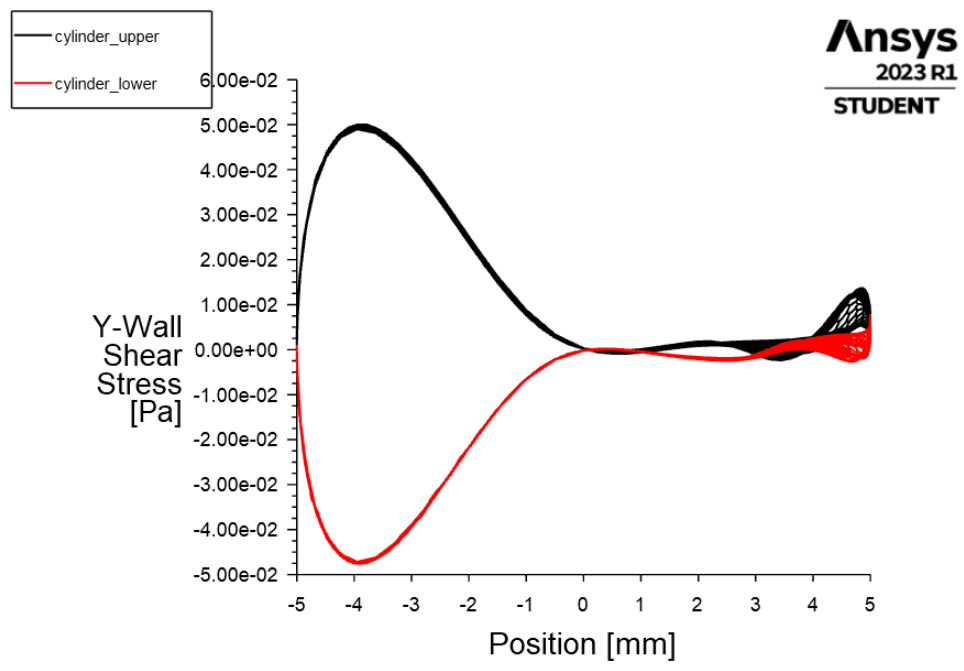
Fig: 1 CAD model for the simulation

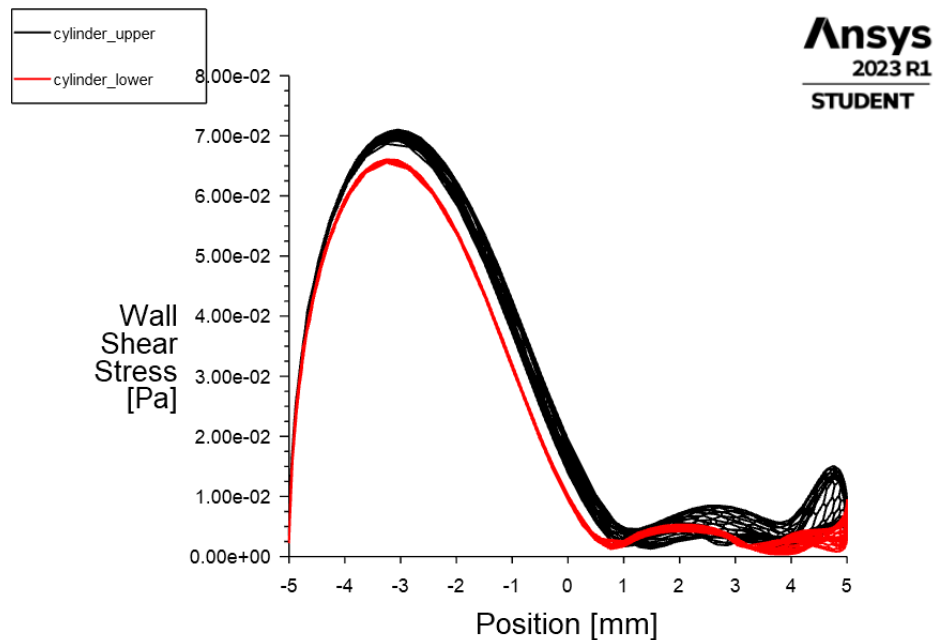
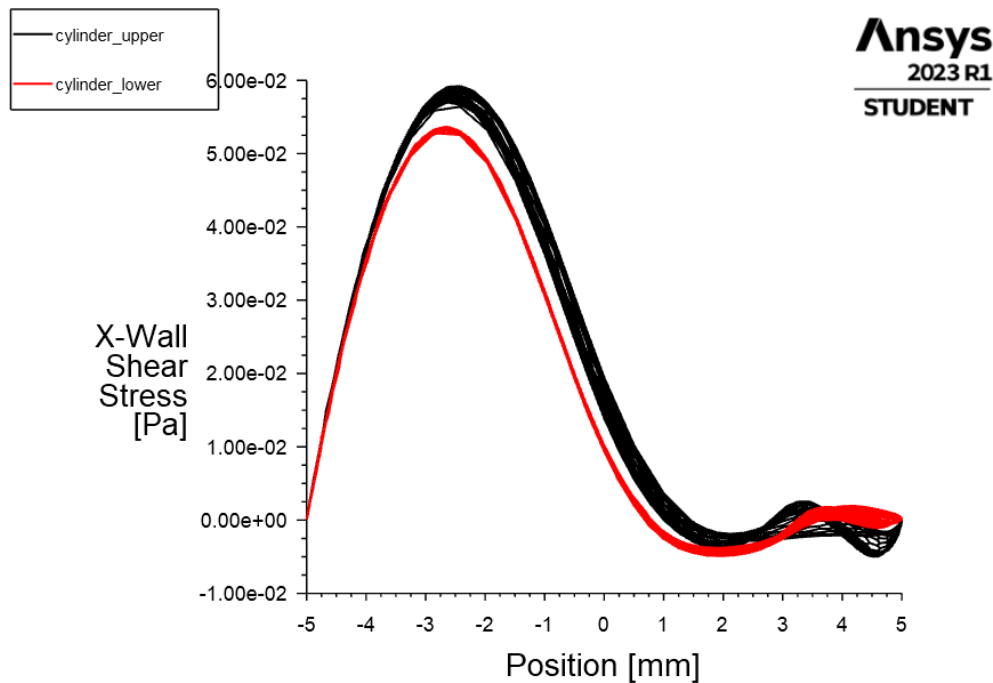
The cad model above shows the computational domain with a circular cylinder placed at the origin. Three separate regions are drawn inside the computational domain which act as BOIs for the mesh refinement.

Meshing -



Solver -





For the post-processing, CFD-post is the best possible option, as the post-processing capabilities of PyFluent is not fully developed currently, as it can be seen from the above plots.

However the purpose of this exercise is to automate the ANSYS Meshing and Fluent solver workflow. The result interpretation can be done using CFD-Post or ParaView.