

Fig. 1

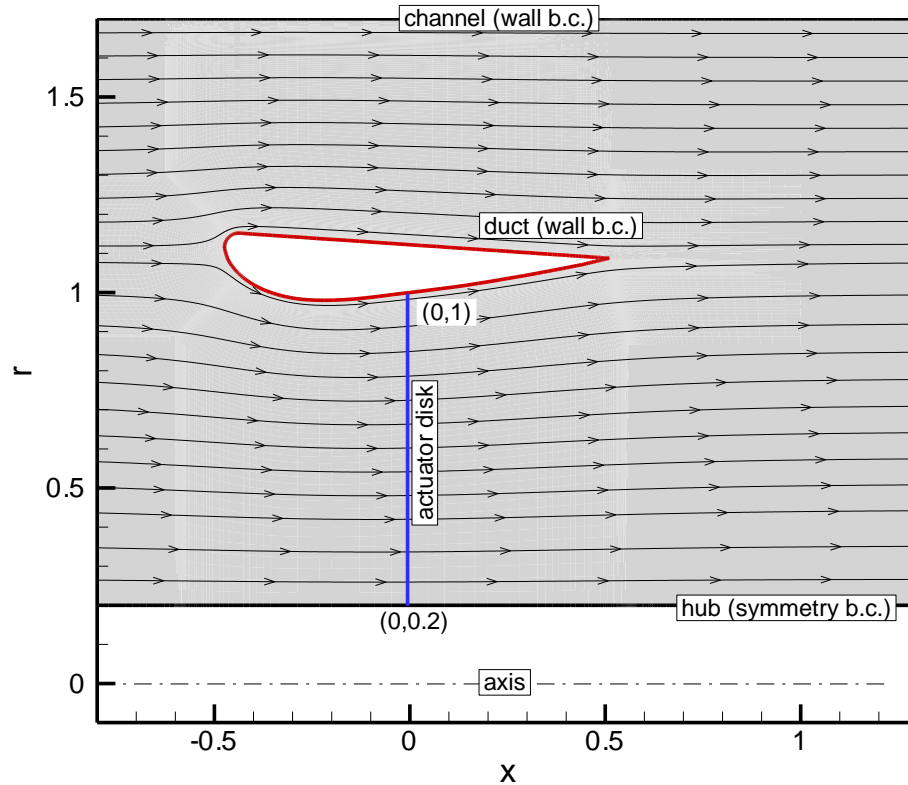
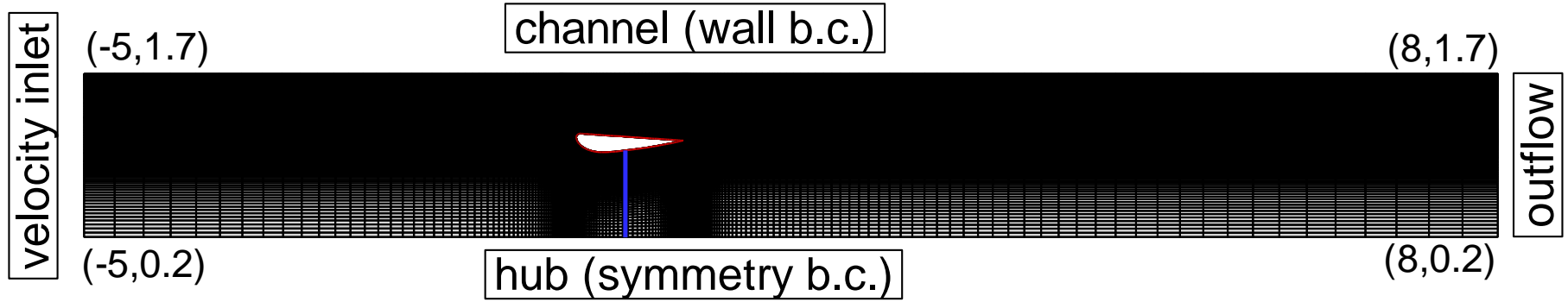
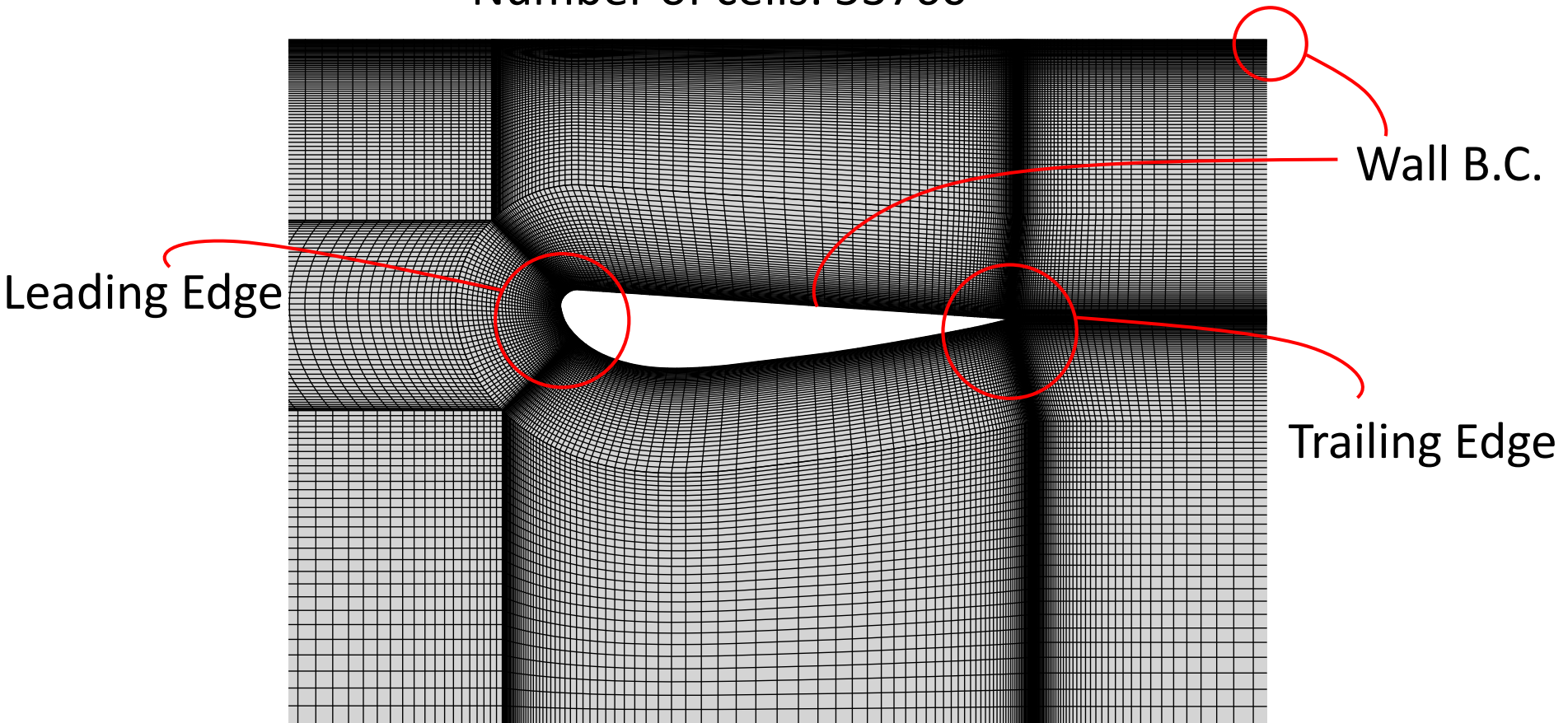


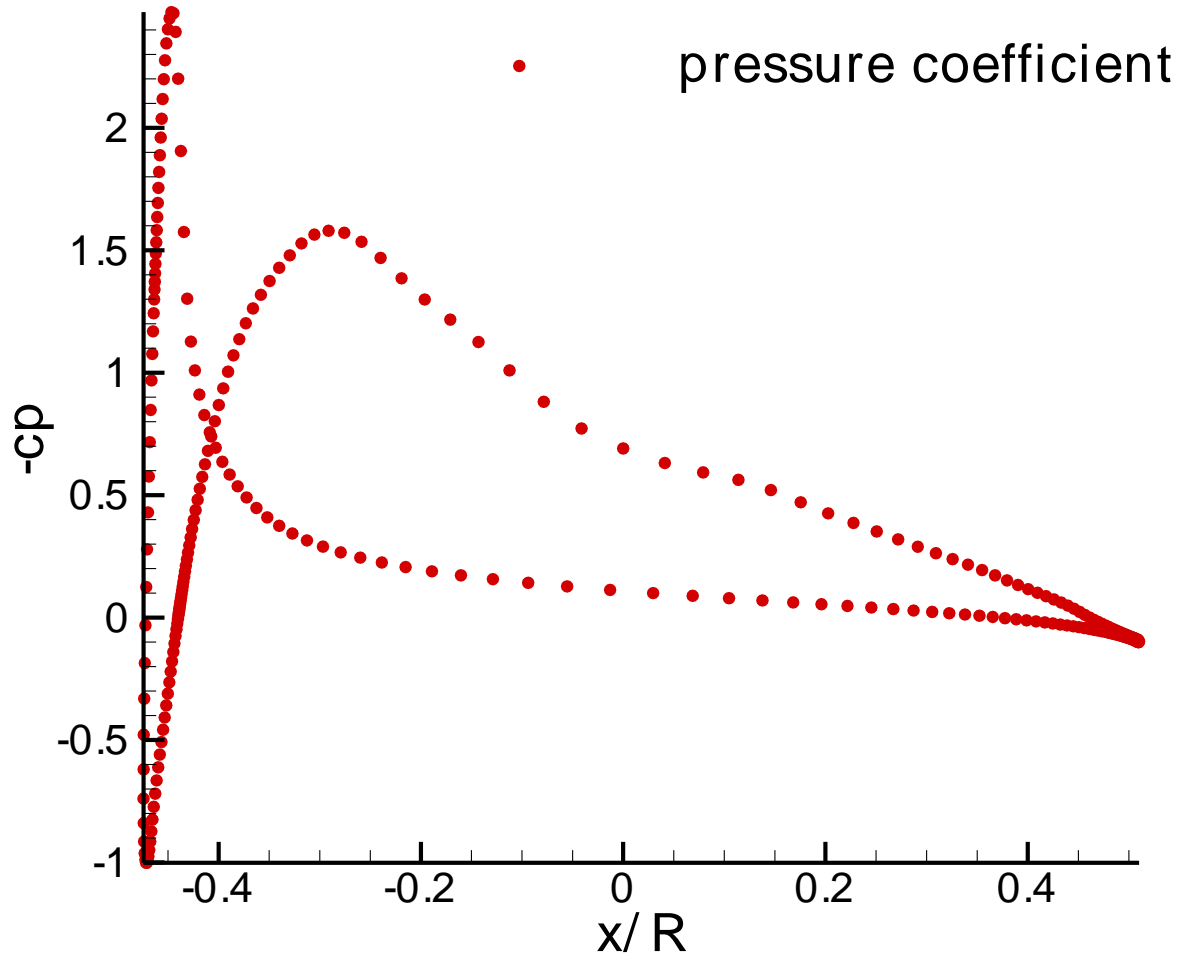
Fig. 2

GRID used in ANSYS/Fluent

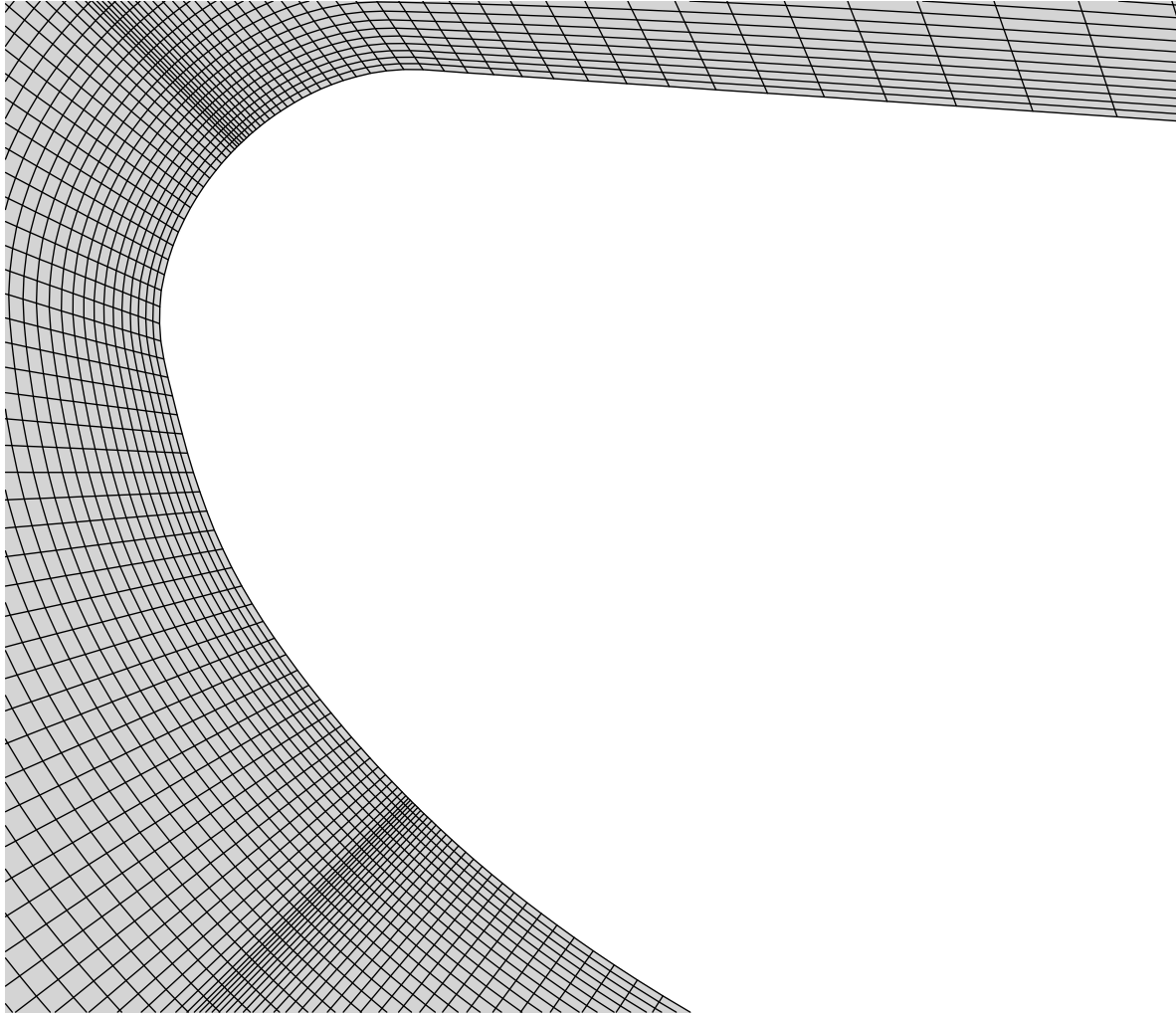


Number of cells: 53760

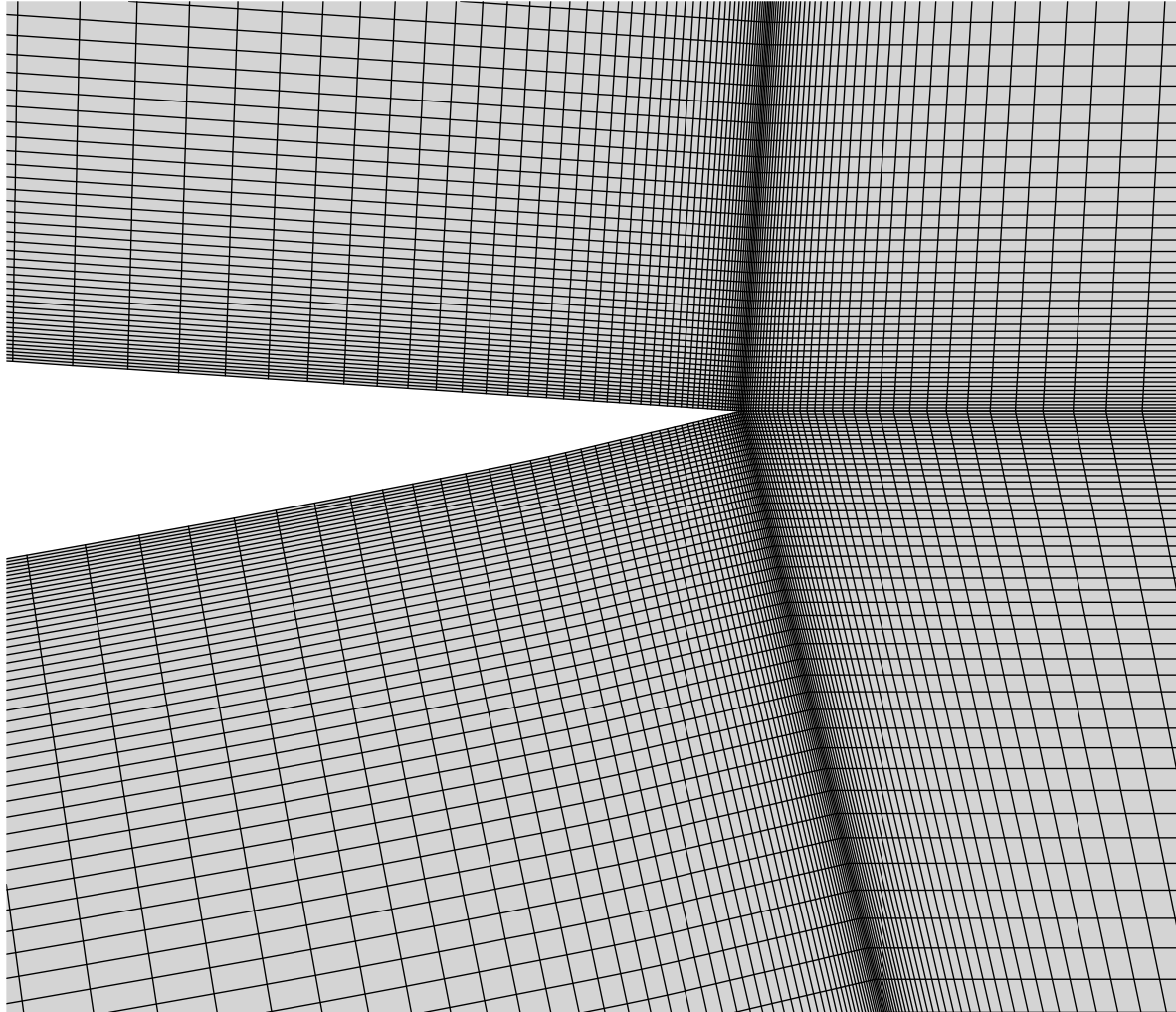




We use finer mesh where the gradient of velocity/pressure is expected to very large (near the leading edge), near walls (to resolve the boundary layers), or near the trailing edge and the wake (to resolve the shed vorticity distribution).



Other rules: try to avoid abrupt change in mesh size, especially near the duct surface



... and try to make the mesh as perpendicular to the duct surface as possible.