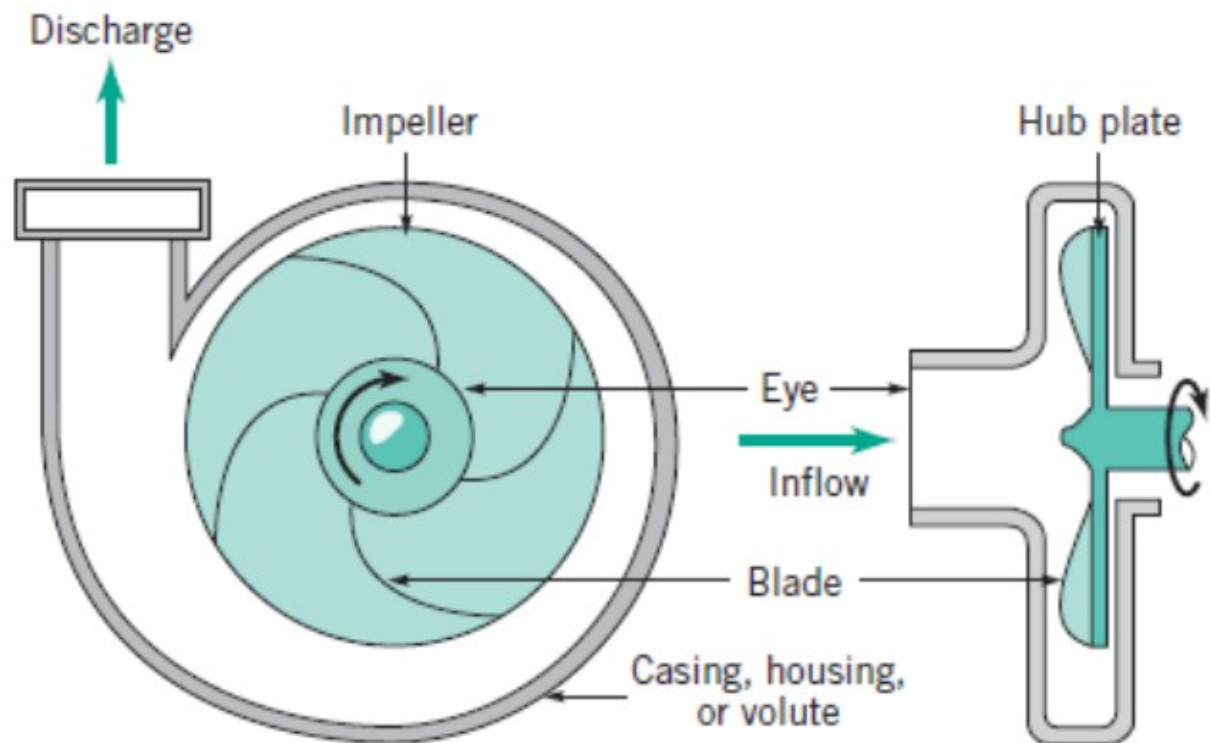


CFD Analysis of a Centrifugal Pump

Aashay Tinaikar, Project Research Assistant, OpenFOAM team, FOSSEE Project, IIT Bombay

To simulate and analyse fluid flow inside a centrifugal pump using opensource softwares OpenFOAM, Gmsh and Salome.

Also to compute and comparing pump efficiency at various operating conditions.



Solver : simpleFoam
Meshing tool : snappyHexMesh