

CFD ANALYSIS ON CENTRIFUGAL PUMP

ABSTRACT

Centrifugal pumps convert rotational energy, often from a motor, to energy in a moving fluid. Centrifugal Pumps have numerous applications in industries including agriculture, municipal (water and wastewater plants), industrial, power generation plants, petroleum, mining, chemical, pharmaceutical and many others.

Computational Fluid Dynamics is most commonly used tool for simulation and analysis. During the present simulation the 3-D modelling of centrifugal pump was done in CATIA V5 and imported to ANSYS Fluid Flow (Fluent). After naming the model's parts in design modular and meshing the same, the analysis was continued in setup modular of Fluent. In the viscous k-epsilon model, by selecting the working fluid as water the simulation was carried out. By fixing the input parameters: Magnitude of Inlet Velocity, Outlet Pressure, Angular Velocity of Impeller and varying direction of Inlet Velocity, calculations are done in CFD post processor by selecting desired number of time steps and iterations. The pressure head variations can easily be observed in contours generated in Fluent results modular.

Basic idea of analysis is to study the variation of the pressure head by varying direction of Inlet Velocity and by fixing the Inlet Velocity, Outlet Pressure and Angular Velocity of Impeller.

Keywords: Centrifugal Pump, Catia, ANSYS, CFD, Velocity, Pressure and Impeller