**SIMULATION OF CENTRIFUGAL PUMP PERFORCE USING CFD TOOL AND OPTIMIZATION OF THE PUMP FOR THE IMPROVED PERFORMANCE**

**ABSTRACT**

The choice of pumps or pumping systems for slurry transport will depend not only on the flow, head required, suction conditions, type of installation and location, as for any other pump application, but also on the slurry flow regime and properties.

Centrifugal pump radial-flow type is the most common in slurry service. A conventional centrifugal pump is designed to handle clear liquids. However when slurries are to be transported the conventional centrifugal pump has to be modified to handle solid liquid mixtures. The modifications incorporated in the pump include enlargement of flow passages to accommodate bigger solid particles, robust impeller with smaller number of vanes, special seals and proper material of construction to ensure longer life. Conventional design method of centrifugal pump are largely based on the application of empirical and semi-empirical rules along with the use of available information in the form of different types of charts and graphs as proposed by successful designers. As the design of centrifugal pump involve a large number of interdependent variables, several other alternative design are possible for same duty.

Computational fluid dynamics (CFD) is being increasingly applied in the design of the centrifugal pumps. 3-D numerical computational fluid dynamics tool can be used for simulation of the flow field characteristics inside the turbo machinery. Numerical simulation makes it possible to visualize the flow condition inside a centrifugal pump, and provides the valuable hydraulic design information of the centrifugal pumps.

Present work is aimed to analyze the pressure and velocity distribution inside the pump passage and evaluate the pump performance using the Fluent, a computational fluid dynamics simulation tool. A numerical model of an impeller and casing has been generated and the complex internal pressure and velocity distribution are investigated by using the fluent computational code. . Pressure and velocity distribution inside impeller of the centrifugal pump has direct influence due to change flow passage