Performance analysis of Multistage Boiler Feed Pump

Kalyani Bangari[1,*], Dynampally Pavitran[2], Dr. G. Raghavender Rao[3], A Sri Kalyan[4]

[1]Deputy Manager, Heat Transfer & Fluid Flow, BHEL Corporate R&D, Hyderabad, India

[2]Senior Manager, Heat Transfer & Fluid Flow, BHEL Corporate R&D, Hyderabad, India

[3]Additional General Manager, Heat Transfer & Fluid Flow, BHEL Corporate R&D, Hyderabad, India

[4]Deputy Manager, Pumps Engineering, HPEP BHEL RCPuram, Hyderabad, India

([a]Corresponding author, e-mail: kalyanib@bhel.in)

Abstract:

In the present times it would be inevitable to solve the fluid mechanics problems without the use of Computational Fluid Dynamics. It produces quantitative predictions of fluid-flow phenomena based on the conservation laws namely conservation of mass, momentum, and energy that govern fluid motion. Computational Fluid Dynamics is widely being accepted as a reliable engineering design tool across the spectrum of industries, as it considerably reduces the design cycle time and enables designer to have a deeper insight into flow physics through flow visualization. The results obtained from CFD simulations is mostly sensitive to grid, boundary conditions and turbulence models chosen. Over the years many numerical schemes, grid generation techniques and turbulence models have come up for resolving the quantitative & qualitative flow features in the fluid flow. But till date there is no generalized numerical scheme and turbulence model which can tackle any given fluid flow conditions. So, depending on how accurate results are required, a particular numerical scheme and turbulence model is chosen.

CFD analysis of Pumps is extensively carried out in the Pump design and development phase. Multistage pumps are generally used for high head operations. In these pumps fluid flows through several impellers fitted in series. The internal flow in multistage pump is complex, especially the qualitative and quantitative characteristics of different stages are quite different at off design conditions. For such problems an appropriate grid strategy with proper near wall resolution, higher order discretization schemes and a robust and reliable turbulence model has to be employed. Generally, in Industry a segregated strategy of meshing for different components with Interfaces,

second order discretization scheme and two equation based Reynolds Averaged Navier Stokes (RANS) turbulence models are used.

In this study a detailed numerical parameter analysis and their effects on the various quantitative and qualitative flow features are studied in a six stage multistage boiler feed pump. The calculated performance parameters through numerical simulations are validated by direct comparison with tested data. Commercial CFD package ANSYS CFX 22.0 was used for the numerical simulations.