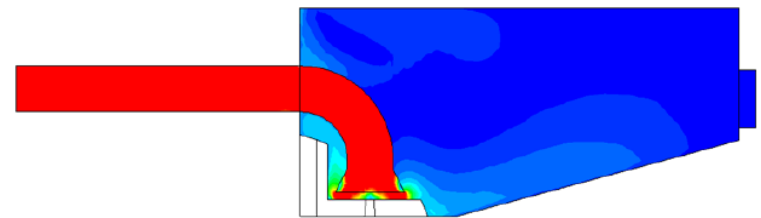
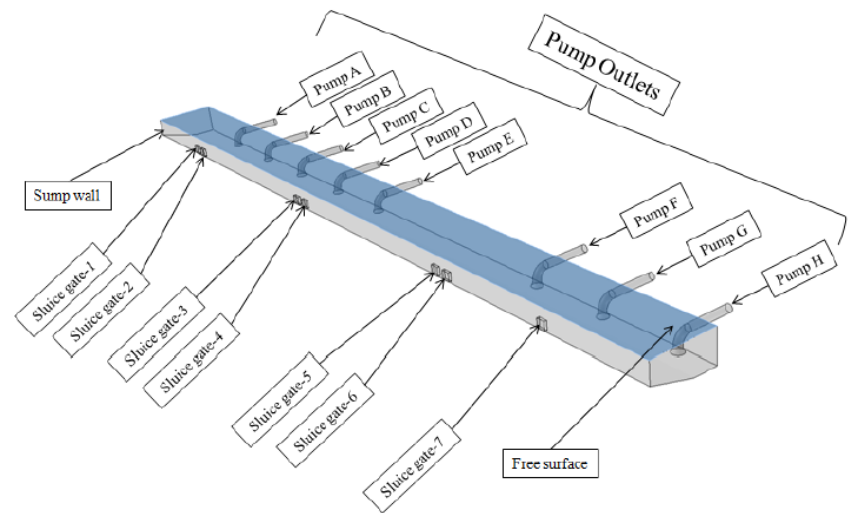
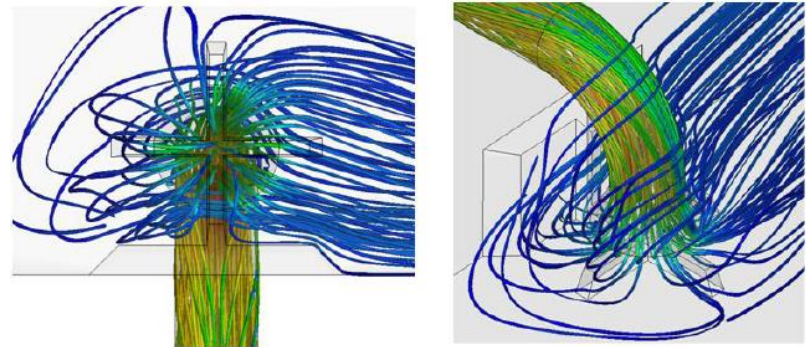


**INTRODUCTION**

In heavy engineering industry, sumps are used to store & supply liquid to the pumps. It is very important that pump suction should receive uniform, vortex free & swirl free flow from the sump, so that pump can perform efficiently at its operating point. Sump containing the multiple number of components make it more complex system to study & judge. CFD helps in understanding the sump performance by simulating all complexities within the sump & also helps in improving the sump performance by carrying out various design modification iterations within the sump.

helps in reducing swirl angle at pump suction. Also CFD helps in determining sump performance during cross flow within sump under various pump & sluice gate working combinations.


**Velocity Plot**

**Pathlines**
**CHALLENGES**

- Proper consideration of resistance for various components like screens and strainers.
- Design modifications, if any required for acceptable performance of Pump Sump as per Hydraulic Institute Standards (HIS).

**THE SOLUTION**

Flow analysis of Pump-Sump is quite challenging due to consideration of various components like multiple number of pumps, sluice gates, raking-screens, traveling screens & strainers which makes the Sump system highly complex to study & judge. CFD helps in identifying the formation of vortex & swirl below bell-mouth, which reduces pump performance by increasing swirl angle at pump suction. After identifying all above issues sump design modified to improve flow pattern within sump & below the bell-mouth which further

**BENEFITS**

- For various pump & sluice gate working combinations, sump performance at pump suction is predicted beforehand.
- Quick turnaround time to high swirl angle problem at pump suction.
- Cost effective solution for reduced number of trials with improved Sump performance.