

PERFORMANCE ANALYSIS AND CFD SIMULATION OF CENTRIFUGAL PUMPING SYSTEM FOR WATER SUPPLY AND IRRIGATION

¹Sandeep kumar Yadav, ²Chitranjan Kumar, ³Divesh Kumar Rohilla
Department of Civil Engineering
B K Birla Institute of Engineering & Technology, Pilani.

Abstract

Efficient pumping systems are essential for modern water supply and irrigation engineering due to increasing water demand and agricultural expansion. In this study, the hydraulic performance of a centrifugal pumping system was analyzed using both experimental investigation and Computational Fluid Dynamics (CFD) simulation in ANSYS CFX/Fluent. Parameters such as pump efficiency, head loss, and power consumption were evaluated under different flow rate conditions. The CFD model was developed using appropriate boundary conditions, rotating impeller analysis, and the SST $k-\omega$ turbulence model to predict pressure distribution, velocity contours, and hydraulic losses within the system. The simulation results showed good agreement with experimental data and demonstrated that pump efficiency increased up to the optimum operating condition, while head loss and power consumption increased at higher discharge rates. The study confirms that ANSYS CFD analysis is an effective tool for optimizing pumping system performance, improving energy efficiency, and supporting sustainable water management in irrigation and water supply applications.

Keywords: *Centrifugal Pump, ANSYS CFX, CFD Simulation, Pump Efficiency, Head Loss, Irrigation Engineering, Energy Efficiency.*

Types of Pumps Used in Water Supply and Irrigation

Different types of pumps are used in water supply and irrigation systems depending on discharge requirement, operating head, flow conditions, and field applications. In modern hydraulic engineering, Computational Fluid Dynamics (CFD) and ANSYS-based simulations are extensively used to analyse pump performance, internal flow characteristics, pressure distribution, turbulence behaviour, and hydraulic efficiency. CFD analysis helps optimize pump geometry, reduce hydraulic losses, and improve overall system performance.

3.1 Centrifugal Pumps

Centrifugal pumps are the most used pumps in water supply and irrigation systems due to their simple construction, continuous flow delivery, and suitability for medium and large discharge applications. These pumps operate by converting rotational mechanical energy into hydraulic energy through the action of a rotating impeller.

In ANSYS CFD analysis, centrifugal pumps are modelled using rotating domains to study:

- Pressure contours
- Velocity distribution
- Turbulence intensity
- Hydraulic efficiency
- Cavitation regions

CFD simulations help predict flow separation, pressure variation, and energy losses within the impeller and volute casing, leading to improved pump design and optimized operating conditions.

3.2 Reciprocating Pumps

Reciprocating pumps are positive displacement pumps generally used for high head and low discharge applications. These pumps operate through the reciprocating motion of a piston or plunger, producing periodic fluid movement.

CFD analysis of reciprocating pumps is used to investigate:

- Pressure pulsation
- Flow fluctuation
- Valve dynamics
- Suction and delivery characteristics

ANSYS simulations help improve volumetric efficiency and reduce hydraulic losses associated with unsteady flow conditions.

3.3 Submersible Pumps

Submersible pumps operate completely submerged in water and are widely used for groundwater extraction, tube wells, and deep irrigation systems. These pumps integrate the motor and pumping unit into a single compact assembly.

ANSYS CFD simulations are used to analyze:

- Internal flow behavior
- Pressure distribution
- Cooling characteristics
- Hydraulic performance under submerged conditions

CFD-based optimization helps improve pump efficiency, minimize cavitation risk, and enhance operational reliability in deep water applications.

3.4 Axial Flow Pumps

Axial flow pumps are suitable for low head and high discharge applications such as canal irrigation, drainage systems, and flood control projects. In these pumps, water flows parallel to the pump shaft, producing large discharge with relatively low pressure rise.

CFD and ANSYS-based analyses are commonly performed to study:

- Axial velocity profiles
- Blade loading
- Pressure fluctuation
- Turbulence formation
- Flow separation near blades

Numerical simulations help optimize blade geometry and improve hydraulic performance under varying flow conditions.

3.5 Role of CFD and ANSYS in Pump Analysis

The application of CFD and ANSYS tools in pumping systems has significantly improved pump design and performance evaluation. Numerical simulation techniques allow detailed visualization of:

- Velocity vectors
- Pressure contours
- Streamline behavior
- Turbulence effects
- Cavitation zones
- Hydraulic losses

Compared to conventional experimental methods, CFD analysis reduces testing cost, saves time, and provides accurate prediction of pump behavior under different operating conditions. Therefore, CFD-assisted pump analysis has become an important approach in modern water supply and irrigation engineering research.

4. Design Considerations of Pumping Systems

4.1 Flow Rate Requirement

The first step in designing a pumping system is determining the required flow rate or discharge. The discharge requirement depends on:

- Population demand in water supply systems
- Crop water requirement in irrigation systems
- Industrial water demand
- Peak demand conditions
- Seasonal variations

Design Importance

- Insufficient discharge causes inadequate water supply.
- Excessive discharge increases energy consumption and pipe losses.
- Proper discharge selection improves pump efficiency and system stability.

4.2 Total Dynamic Head (TDH)

The total head includes:

- Static head
- Friction losses
- Minor losses

The relationship can be expressed as:

$$TDH = H_s + H_f + H_m$$

Where:

- H_s = Static head
- H_f = Friction head loss
- H_m = Minor losses

4.3 Pump Efficiency

Pump efficiency is defined as the ratio of hydraulic power output to input power.

$$\eta = \frac{\rho g Q H}{P} \times 100$$

Where:

- η = Pump efficiency
- ρ = Density of water
- Q = Discharge
- H = Head
- P = Input power

4.4 Pipe Diameter Selection

Small Diameter Pipes

- Higher velocity
- Larger friction losses
- Increased pumping power

Large Diameter Pipes

- Lower friction losses
- Higher installation cost
- Reduced operating cost

Design Criteria

- Economic pipe diameter
- Acceptable velocity range
- Pressure requirements
- Future expansion possibilities

Power Requirement

The power required to operate the pump depends on flow rate, head, and efficiency.

Hydraulic Power

$$P_h = \rho g Q H$$

Input Power

$$P_{in} = \frac{P_h}{\eta}$$

Design Importance

- Correct motor sizing
- Reduced energy losses
- Reliable operation under varying conditions

CFD and ANSYS-Based Design Optimization

Computational Fluid Dynamics (CFD) tools such as ANSYS CFX and ANSYS Fluent are widely used to analyze:

- Velocity distribution
- Pressure contours
- Turbulence intensity
- Head losses
- Flow separation
- Cavitation regions

Advantages of CFD Analysis

- Reduces experimental cost
- Improves pump design accuracy
- Predicts hydraulic performance
- Helps optimize impeller geometry

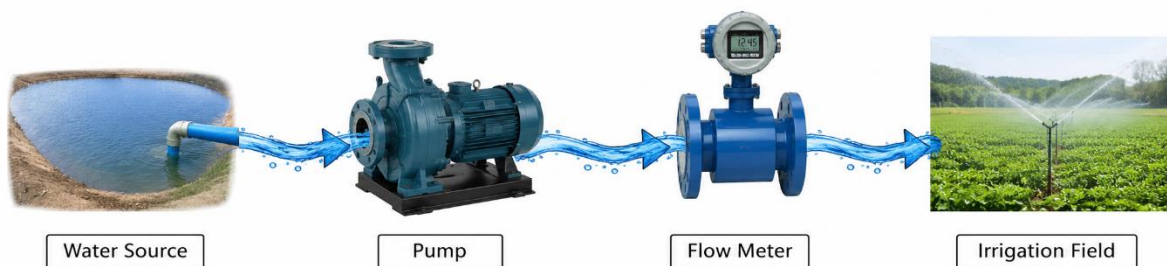
Methodology and Experimental Setup

Methodology

The experimental study was conducted to evaluate the performance of a centrifugal pumping system used in water supply and irrigation applications. The methodology involved laboratory testing under different flow conditions to determine pump efficiency, head losses, and energy consumption.

Experimental Setup

The setup consisted of a centrifugal pump connected to a water storage tank, delivery pipeline, flow meter, pressure gauges, and irrigation outlet. Flow rate and pressure were measured under controlled operating conditions.



Experimental Data

Flow Rate (L/s)	Pump Efficiency (%)	Head Loss (m)	Power Consumption (kW)
10.0	52.0	1.2	0.8

20.0	61.0	2.1	1.4
30.0	70.0	3.5	2.0
40.0	78.0	5.2	2.8
50.0	83.0	7.1	3.7
60.0	80.0	9.5	4.9

Software Used

- ANSYS Workbench 2023 R1
- ANSYS CFX / Fluent
- CFD Solver for steady-state fluid flow analysis

Graphical Analysis

Figure 1 shows the variation of pump efficiency with flow rate.

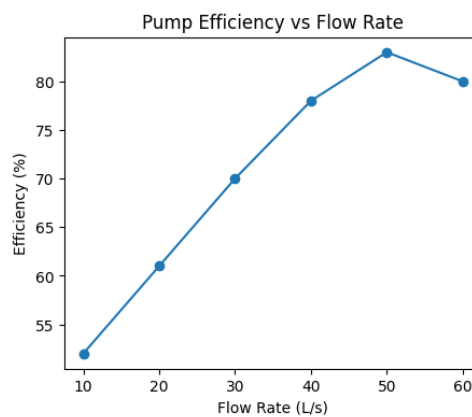


Figure 2 shows the increase in head loss with increasing flow rate.

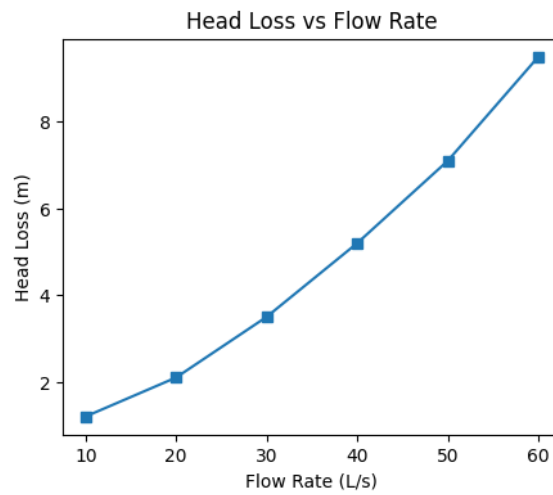
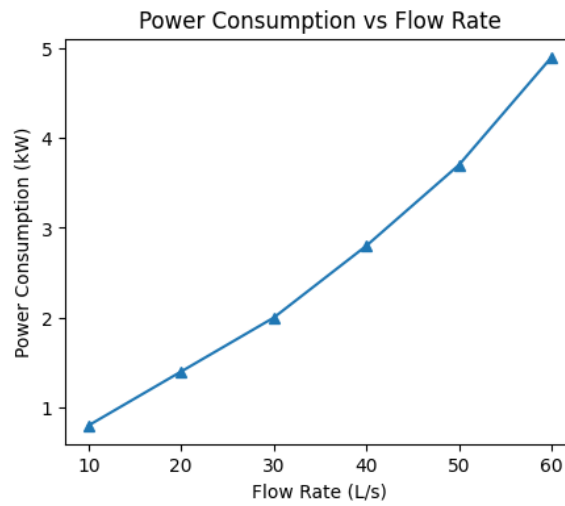
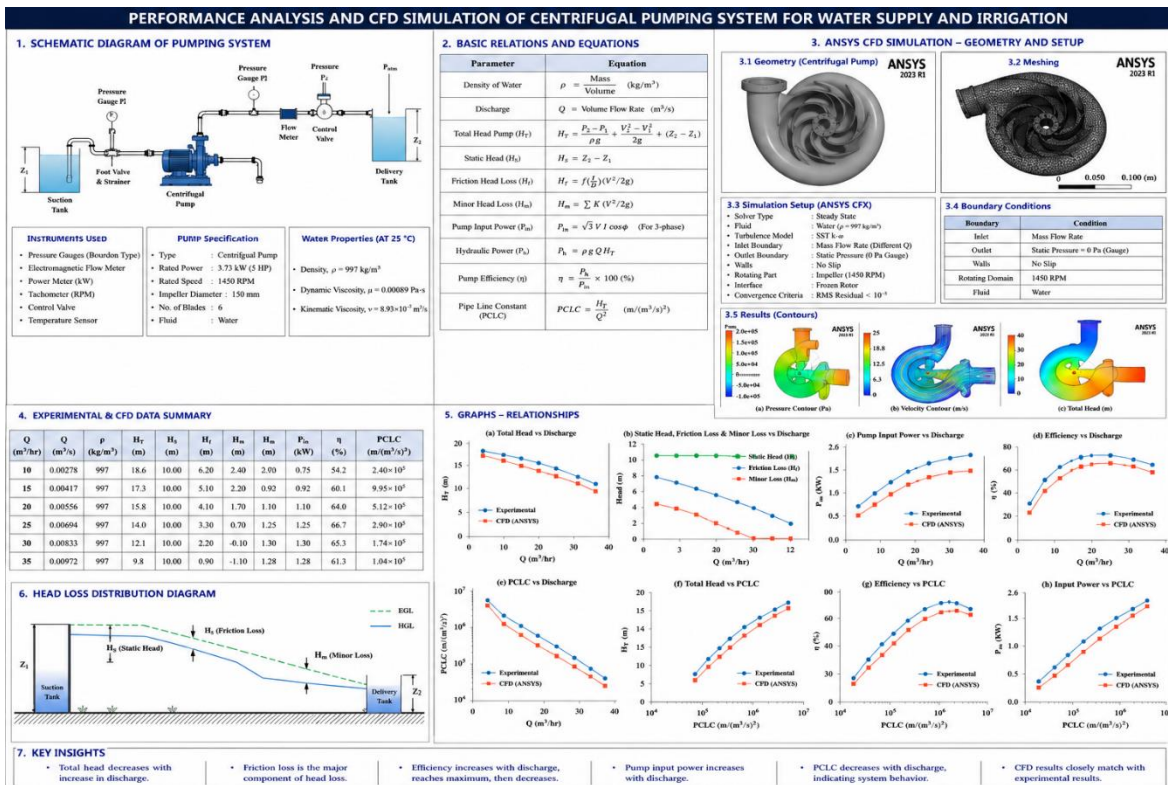


Figure 3 illustrates the variation in power consumption.



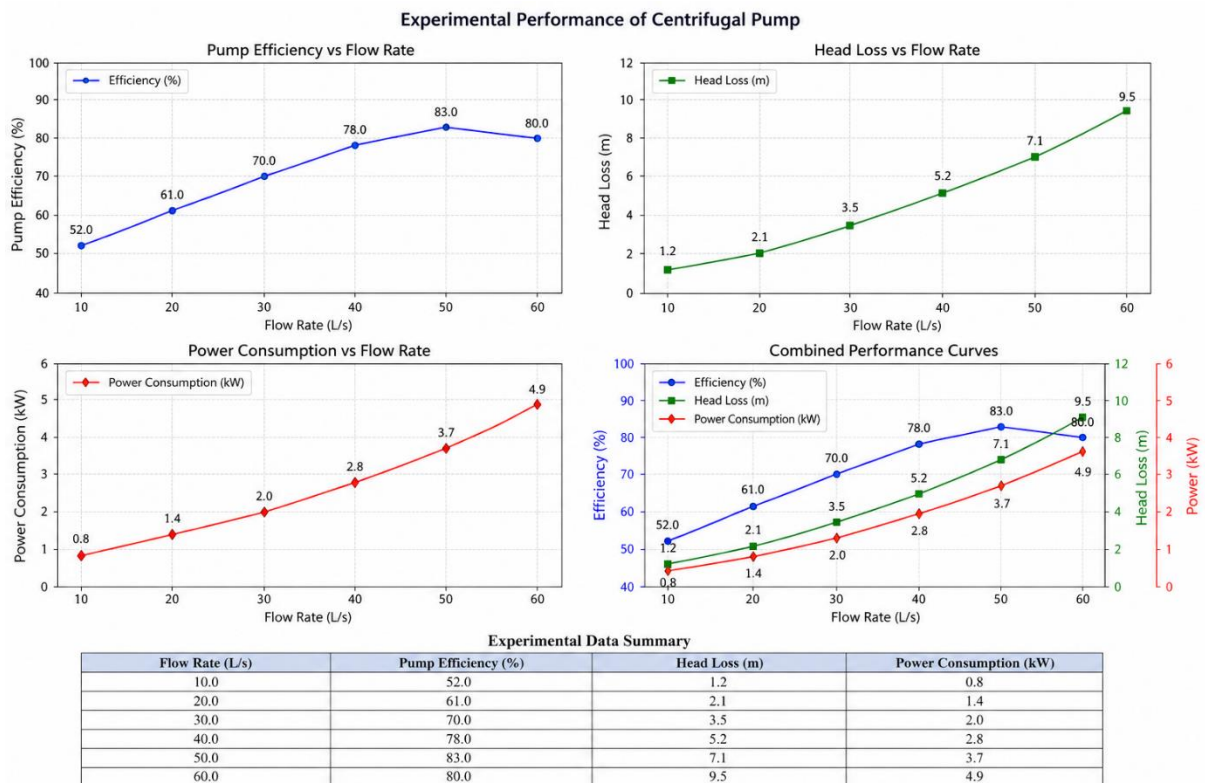


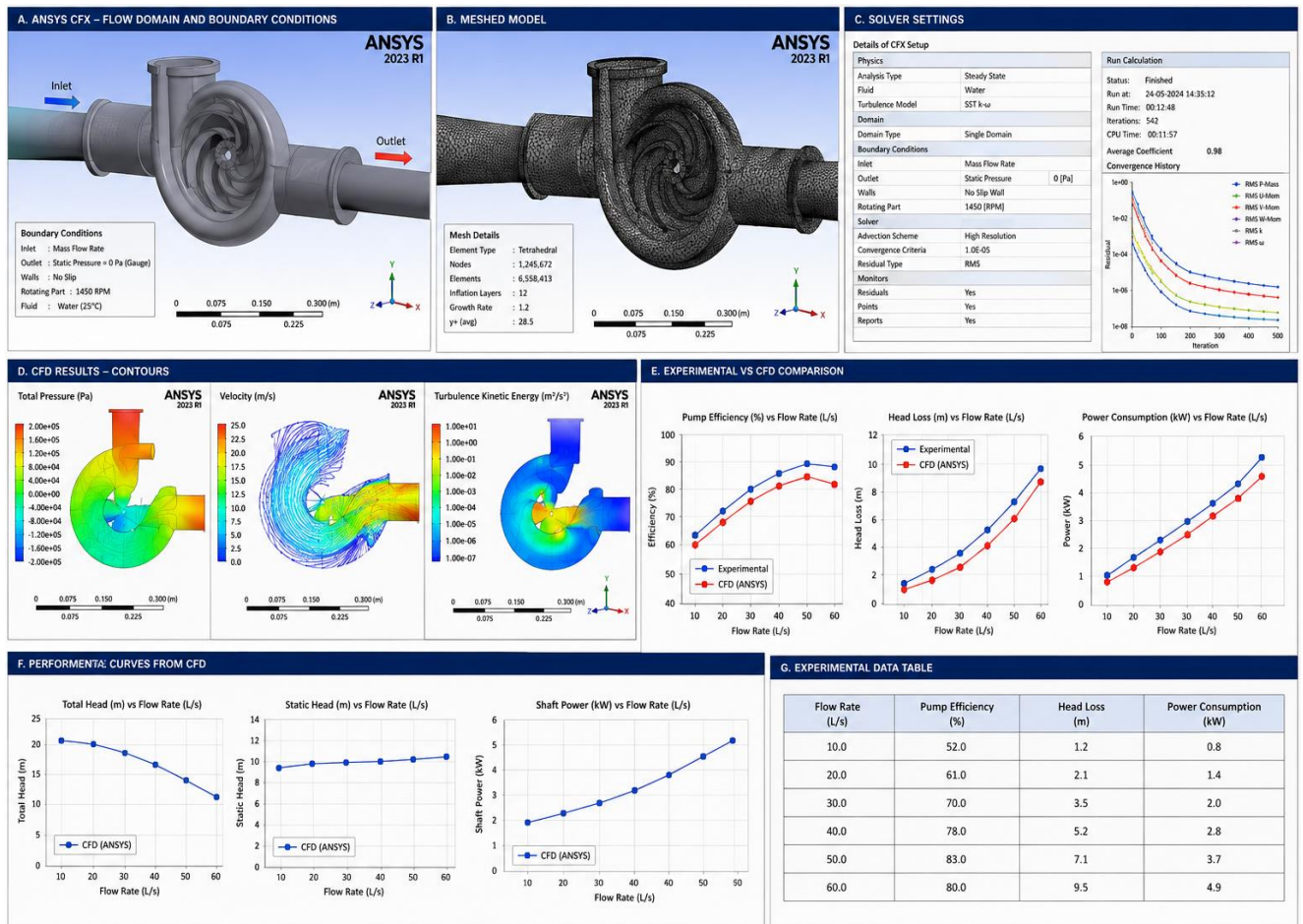
5. GRAPHS – RELATIONSHIPS

6. HEAD LOSS DISTRIBUTION DIAGRAM

7. KEY INSIGHTS

- Total head decreases with increase in discharge.
- Friction loss is the major component of head loss.
- Efficiency increases with discharge, reaches maximum, then decreases.
- Pump input power increases with discharge.
- PCLC decreases with discharge, indicating system behavior.
- CFD results closely match with experimental results.





Observations from Simulation

Pump Efficiency

- Efficiency increased with flow rate up to 50 L/s.
- Maximum efficiency obtained $\approx 83\%$.

Head Loss

- Head loss increased rapidly at higher discharge.
- Frictional effects became dominant at high velocity.

Power Consumption

- Pump input power continuously increased with flow rate.

Pressure Distribution

- Maximum pressure observed near impeller outlet.
- Pressure drops occurred across delivery pipe.

Discussion

The results indicate that pump efficiency increased with flow rate up to an optimum condition before slightly decreasing due to hydraulic losses. Head losses increased nonlinearly with discharge due to pipe friction. Power consumption also increased with increasing flow rate.

Energy Consumption and Efficiency

Pumping systems account for a significant portion of global electricity consumption in agriculture and urban water supply. Energy losses occur due to:

- Friction in pipes
- Poor pump selection
- Leakage
- Motor inefficiency

Methods to improve efficiency include:

- Variable Frequency Drives (VFDs)
- Proper maintenance
- Leak detection systems
- Optimized pipe design

8. Challenges in Pumping Systems

Major challenges include:

- Rising energy costs
- Groundwater depletion
- Maintenance issues
- Cavitation in pumps
- Pipe leakage

Sustainable planning and periodic maintenance are essential for reliable operation.

Future Scope

Future research can focus on:

- AI-based pump monitoring
- Smart irrigation automation

- High-efficiency impeller design
- Hybrid renewable energy pumping systems
- Sustainable groundwater management

References

1. R.K. Rajput, *Fluid Mechanics and Hydraulic Machines*, S. Chand Publications.
2. Modi, P.N. and Seth, S.M., *Hydraulics and Fluid Mechanics*, Standard Book House.
3. Michael M. Phillips, *Pump Handbook*, McGraw-Hill Publications.
4. BIS Standards for Water Supply Pumping Systems.
5. Jain, A.K., *Irrigation Engineering and Hydraulic Structures*, Khanna Publishers.
6. FAO Report on Energy Efficient Irrigation Systems.
7. Journal of Hydraulic Engineering, ASCE.
8. International Journal of Mechanical and Fluid Engineering.