Project Report: CFD Analysis of a Centrifugal Pump Using SolidWorks Flow Simulation

Title: CFD Analysis of Total Pressure Variation with Outlet Velocity in a Centrifugal Pump
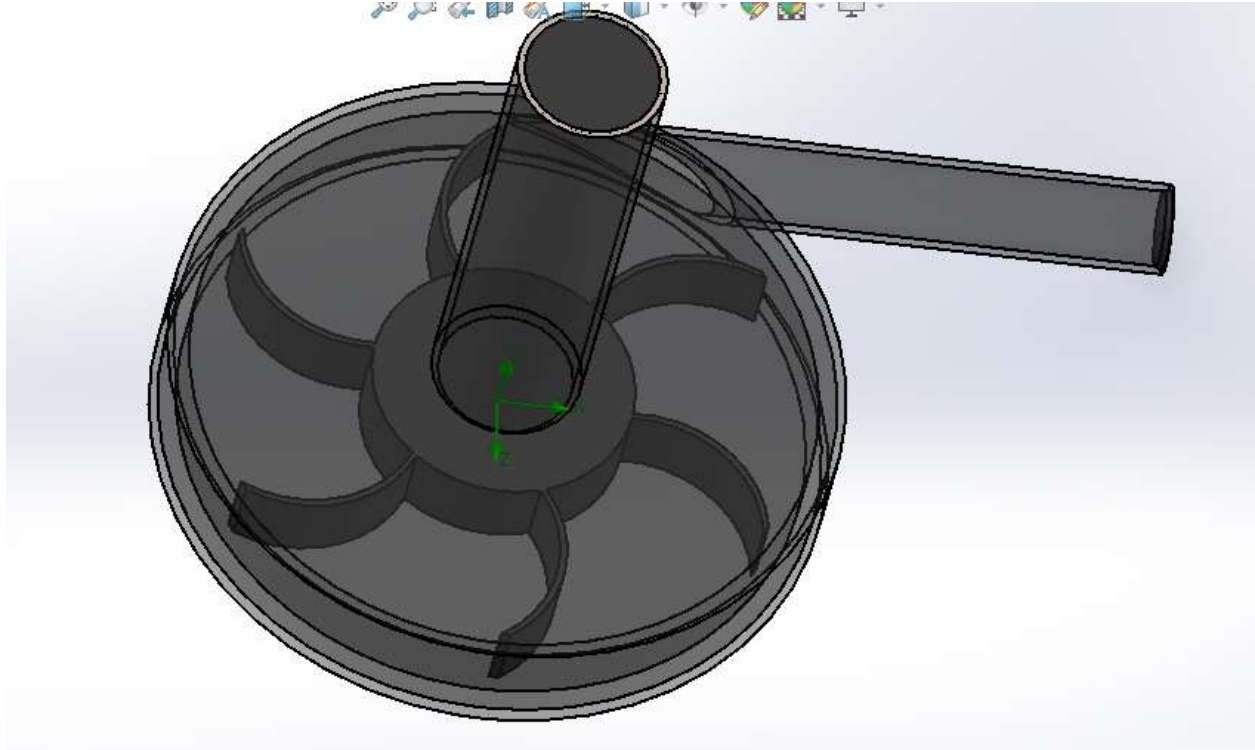**Author:** Arjun singh
Institute: MANIT Bhopal
**Course:** B.Tech Mechanical Engineering – 5th Semester

*. Table of Contents**

Abstract

This project presents a Computational Fluid Dynamics (CFD) analysis of a centrifugal pump using SolidWorks Flow Simulation. The aim is to understand how varying outlet velocities affect the total pressure at the outlet of the pump. A 3D CAD model of a centrifugal pump was created, and internal flow simulation was conducted under steady-state conditions. Parametric analysis was performed by varying the outlet velocity from 10 m/s to 30 m/s. Simulation results showed a non-linear decrease in total pressure with increasing velocity, consistent with expected flow behavior in pump systems
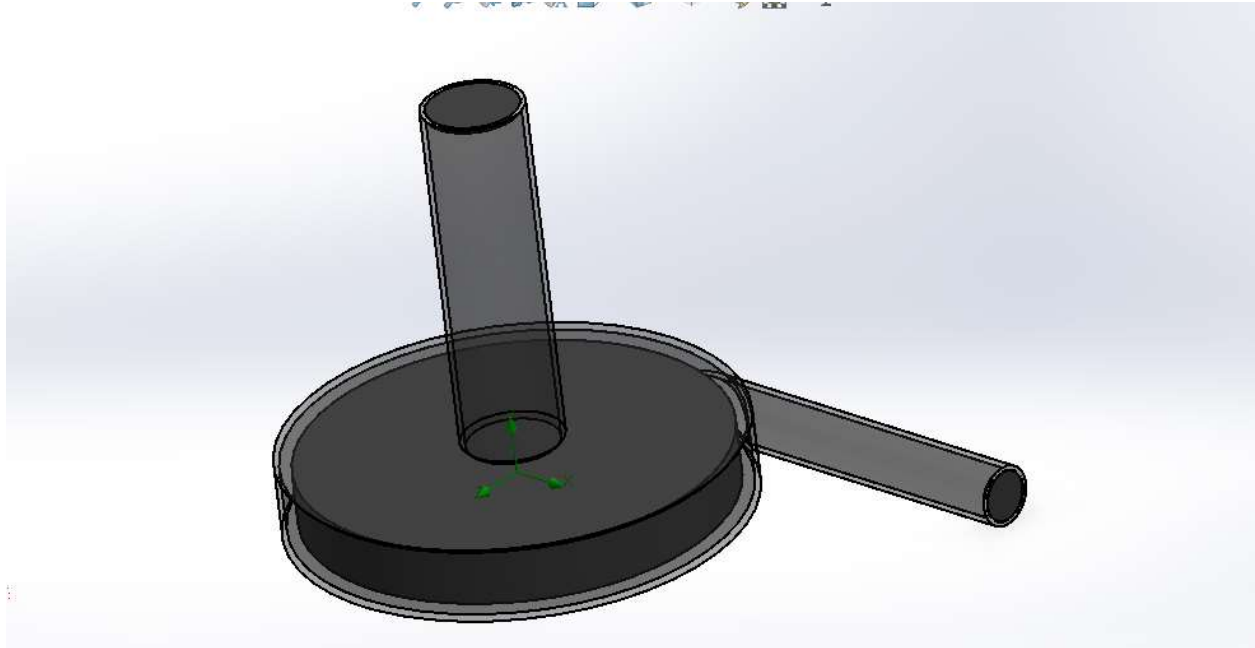
Introduction

A centrifugal pump is a mechanical device used to transport fluids by converting rotational kinetic energy, typically from an electric motor, into hydrodynamic energy of the fluid flow. It is widely used in various industries such as water treatment, chemical processing, oil refining, and HVAC systems due to its efficiency and capability to handle large flow rates.

In a centrifugal pump, fluid enters the impeller axially and is accelerated radially outward due to centrifugal force. This motion increases the fluid's kinetic energy, which is partially converted into pressure energy in the volute casing. Understanding the relationship between outlet velocity and total pressure is vital in pump design, as it affects the system's energy efficiency and reliability.

This study uses CFD simulations to investigate how different outlet velocities influence the total pressure at the pump outlet. The findings can aid in refining design parameters and operational limits for optimal pump performance.

. Objectives

- To design a 3D model of a centrifugal pump using SolidWorks.
- To simulate internal fluid flow using SolidWorks Flow Simulation.
- To conduct a parametric study by varying outlet velocities.
- To analyze and interpret the pressure behavior at the outlet.
- To predict performance trends using regression for higher velocities.

. Geometry and Setup

The centrifugal pump geometry includes a volute casing, an impeller chamber, inlet and outlet ports. The model was created in SolidWorks using accurate dimensions suitable for internal flow simulation.

Geometry Components:

- Impeller (modeled as stationary)
- Inlet: Circular, axial entry
- Outlet: Tangential exit

Boundary Conditions:

- Inlet: Environmental pressure
- Outlet: Velocity (Parametric: 10–30 m/s)
- Walls: No-slip, adiabatic

Fluid Properties:

- Air at standard temperature and pressure (STP)
- Turbulent flow with default k-epsilon model

Simulation Methodology

- Type: Internal, steady-state flow analysis
- Solver: SolidWorks Flow Simulation
- Goals: SG Average Total Pressure at the outlet face
- Parametric study defined using Design Points for outlet velocities
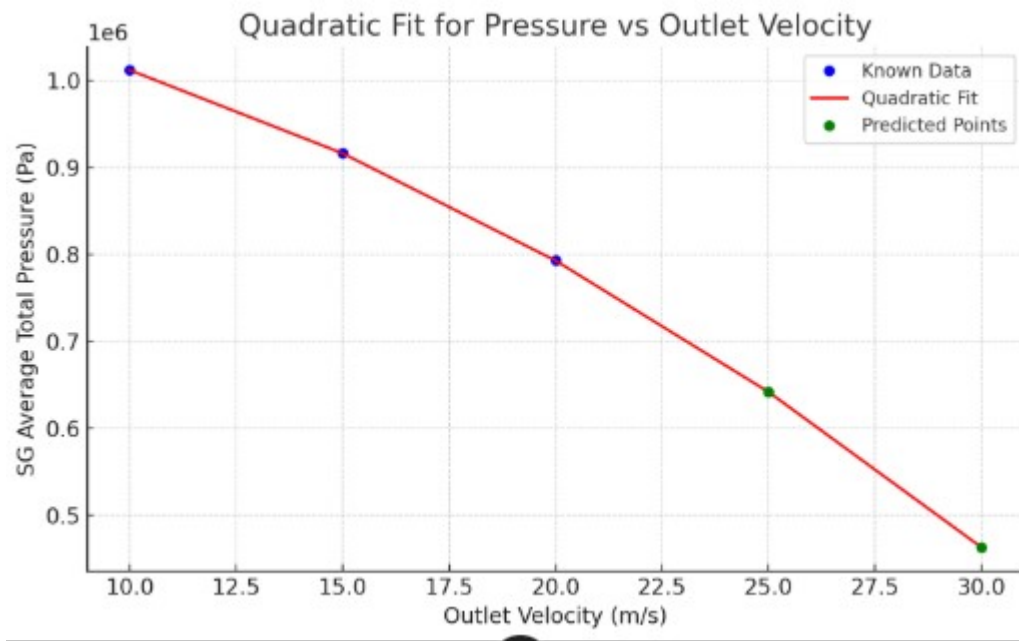- Velocity points: 10m/s, 15m/s, 20 m/s 25m/s, 30m/s
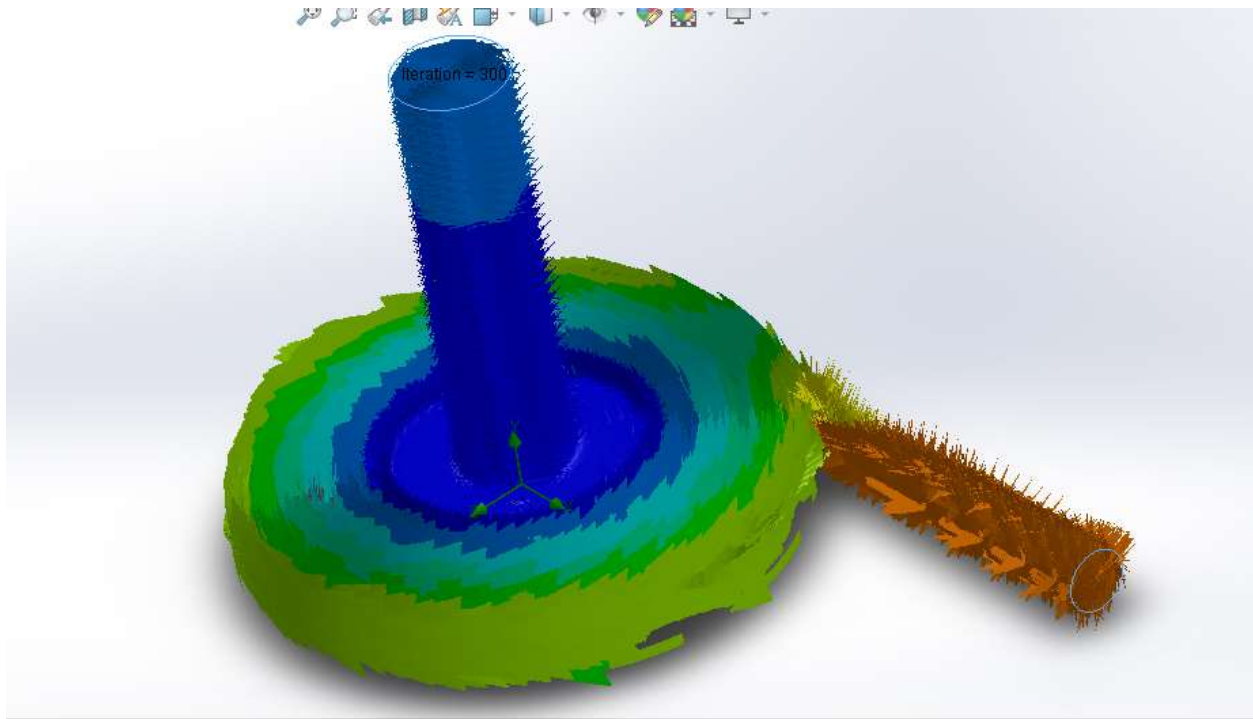
Animation.avi

Results and Discussion

Simulation Data:

Outlet Velocity (m/s)	Total Pressure (Pa)
10	1,011,767.27
15	916,078.63
20	792,736.18
25	759,336.26
30	607,975.16

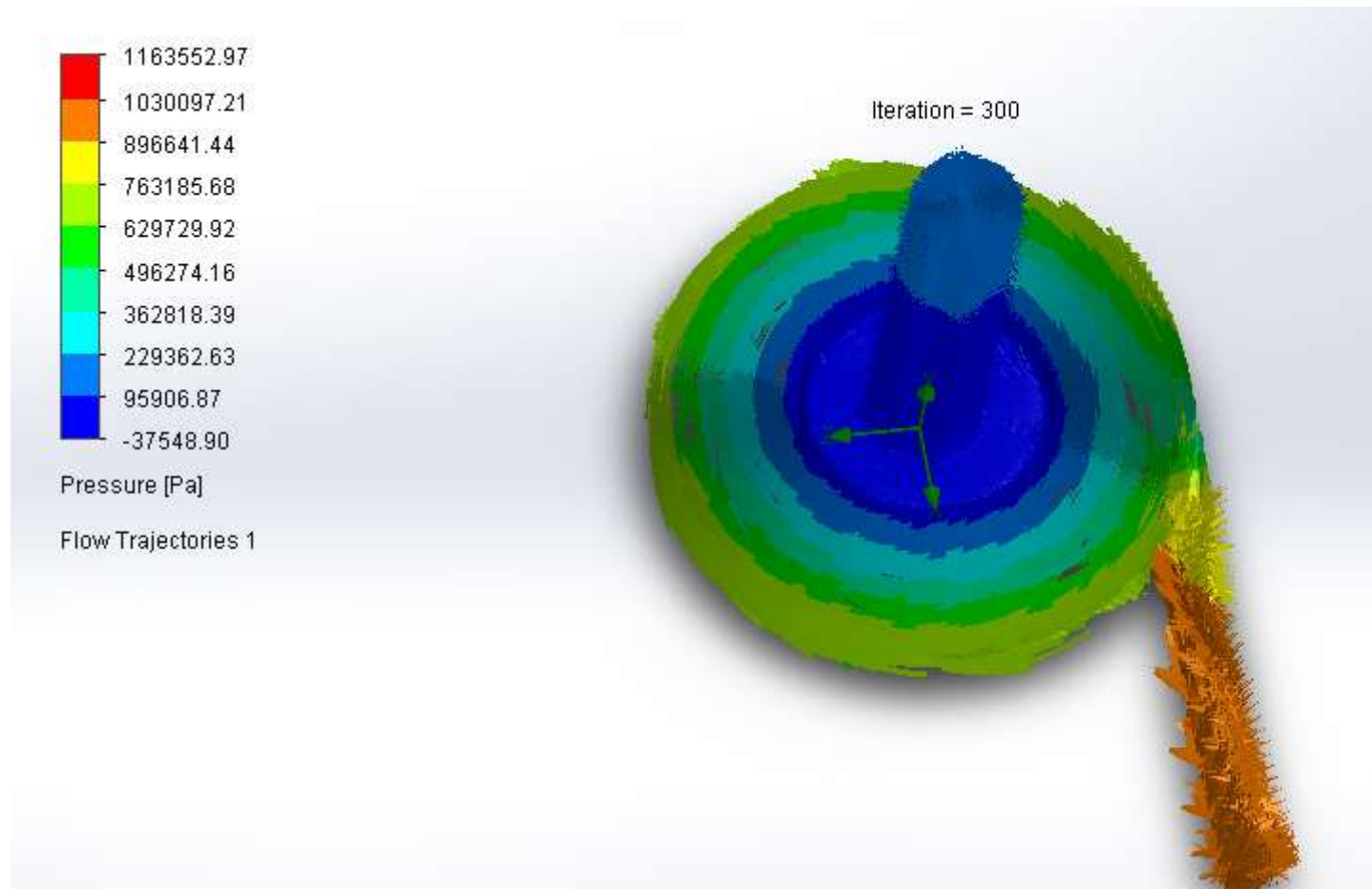**Quadratic Fit for Pressure vs Outlet Velocity**

Observations:

- A clear non-linear decrease in pressure with velocity
- Energy losses and turbulence increase with outlet speed
- Trend fitted using quadratic regression for accurate estimation

.Conclusion

The CFD simulation successfully modeled internal flow in a centrifugal pump. The results showed that outlet velocity significantly affects total pressure, and the trend is non-linear due to turbulence and flow losses. The use of regression to predict untested conditions extended the study's insight. This approach is helpful in pump design for estimating performance without running multiple simulations.

Iteration = 300

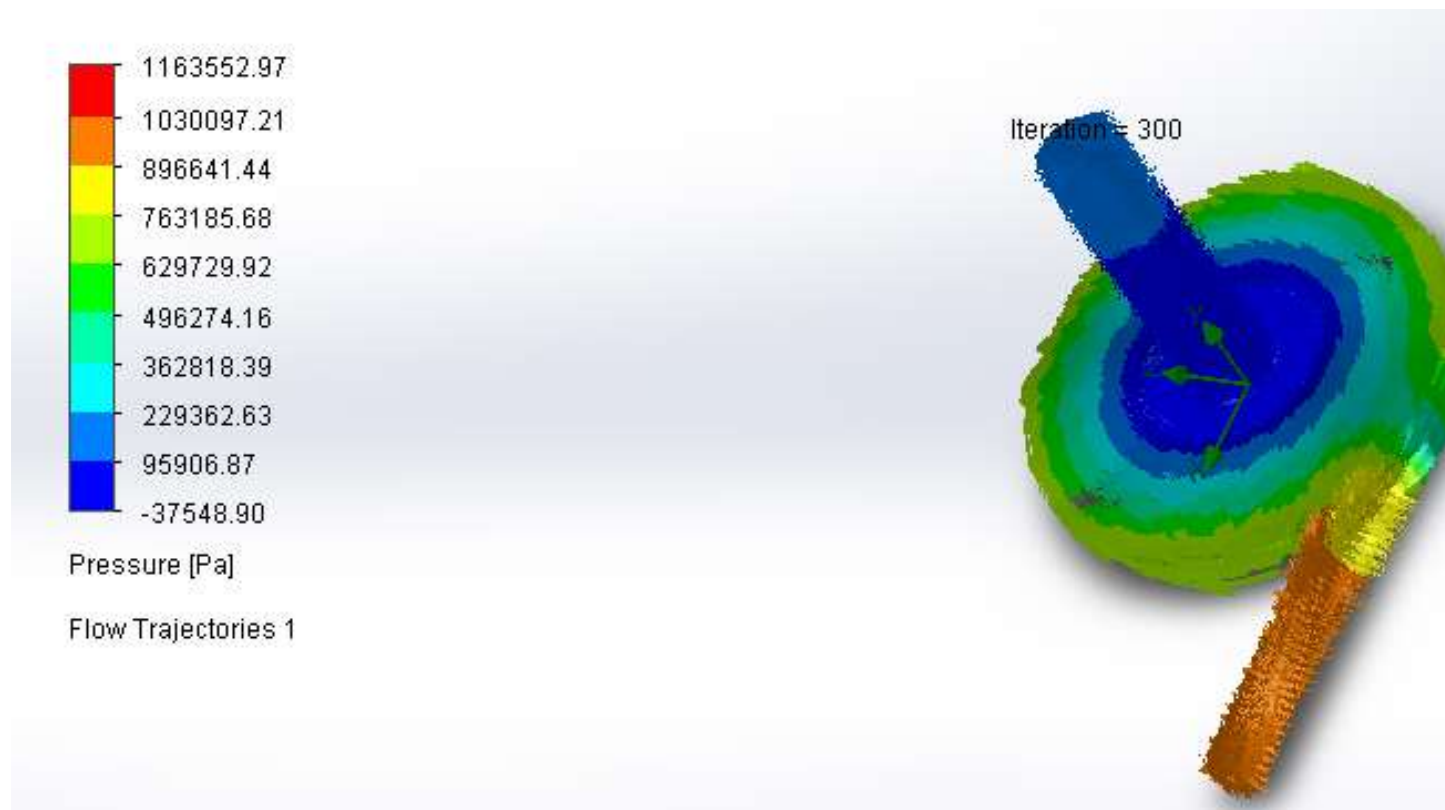Pressure [Pa]

Flow Trajectories 1

. References

1. SolidWorks Flow Simulation Help Files
2. White, F.M. - Fluid Mechanics, McGraw-Hill
3. Engineering Toolbox – Fluid Properties
4. CFD Online Resources

13. Appendix

- Screenshots of geometry and mesh
- Simulation setup dialogs
- Surface goal result logs
- Parametric table screenshots
- Graph export from regression fit

Pressure [Pa]

Flow Trajectories 1

. References

1. SolidWorks Flow Simulation Help Files
2. White, F.M. - Fluid Mechanics, McGraw-Hill
3. Engineering Toolbox – Fluid Properties
4. CFD Online Resources

13. Appendix

- Screenshots of geometry and mesh
- Simulation setup dialogs
- Surface goal result logs
- Parametric table screenshots
- Graph export from regression fit