Design • Analysis • Research

Cooling Pump Design and Testing

1. Introduction

Design, Analysis and Research Corporation (DARcorporation) experience with cooling systems ranges from aviation, automotive and computer electronics. This white paper outlines the design process of a centrifugal pump for a server farm. This process is also applicable to other pump types, such as axial and rotary pumps.

A generic design, to resemble a commercially off-the-shelve (COTS) pump, is used as a baseline. The design objective is to produce a design with the ability to maximize the total head pressure for a specific flow rate of 60 LPM at 5,000 RPM. The flow rate and RPM is selected to narrow the design scope. Note that the design optimization is not limited to total head pressure. Optimization of flow rate and efficiency can be the focus of the design. The generic pump model shown in Figure 1.1.

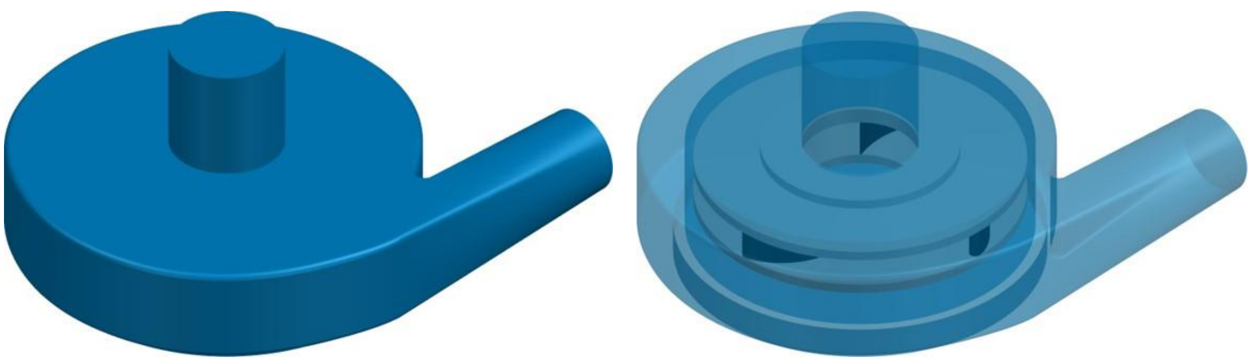

Figure 1.1 Generic Pump

Throughout this paper, impellers are named according to their number of blades and their blade angles. The generic pump impeller is a four bladed design with an inflow and outflow angle of 115 degrees and 160 degrees, 4bl-115160, as shown in Figure 1.2. From here on the generic pump and impeller are referred to as the baseline pump and impeller.

910 East 29th Street, Lawrence, Kansas 66046 USA ● (785) 832-0434
info@darcorp.com ● www.darcorp.com ● www.airplanedesign.aero

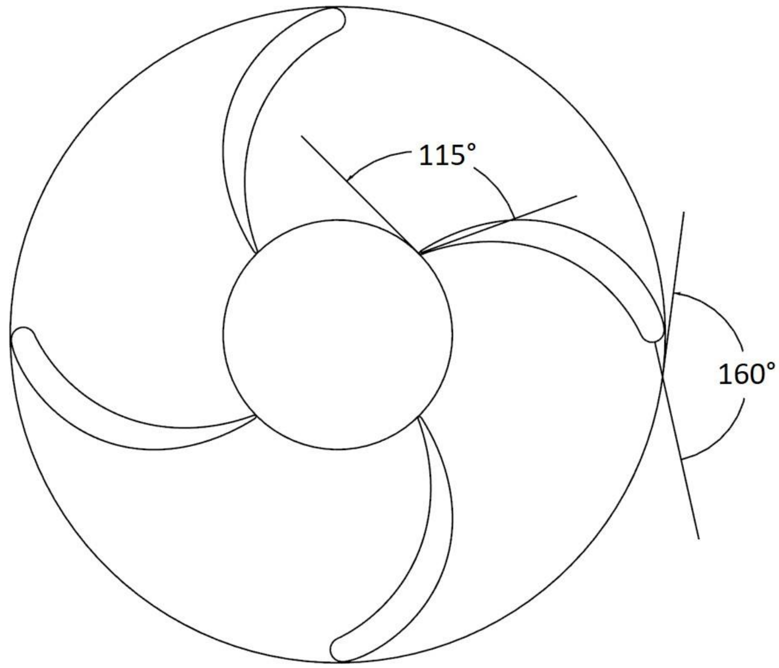

Figure 1.2 Baseline Impeller 4bl-115160

2. Pump Testing

Figure 2.1 shows an overview of the in-house closed loop pump test setup, with the flow direction indicated with blue arrows. The pump is mounted between two tanks with restriction valves on the inlet and outlet. The valves control the flow rate. A flow meter is mounted between the two tanks where the flow is clean, to ensure a stable reading of the flow rate. Pressure sensors are installed at the inlet and outlet to calculate the head pressure of the pump.

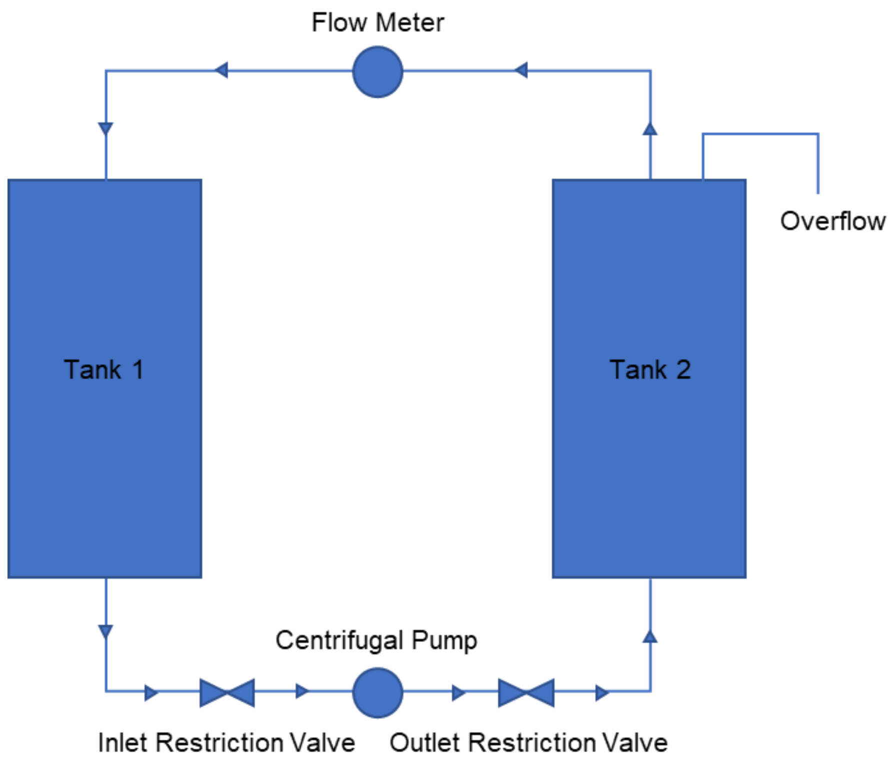

Figure 2.1 Pump Test Setup

The test setup is used to test the performance of a pump, such as head, flow rate, RPM, current and voltage. For this paper, since the demonstrated design is theoretical, there is no test data available.

3. Preliminary Design

An in-house developed tool is used to perform a sensitivity study on the effect of changes to impeller and blade geometry, without the need to run Computational Fluid Dynamic (CFD). Casing optimization is conducted in detailed design using CFD. The analytical model is based on Euler equations and includes a fluid slippage loss model, as well as a volute loss accounting for friction and leakage. Inputs to in model include:

- Volute geometry
- Blade geometry
- Reynolds number

The analytical model is also used to benchmark the baseline pump. This is to capture any performance change due to changes in impeller and blade geometry. Table 3.1 shows the baseline pump performance at 60 LPM and 5,000 RPM.

Table 3.1 Analytical Model Pump Performance

60 LPM at 5,000 RPM	Head Pressure [psi]	Efficiency [~]
4bl-115160 Baseline	48.8	0.23

Head pressure, h is defined as the difference in pressure between the outlet and inlet. Equation (1) is used to determine head pressure.

$$h = P_{oulet} - P_{inlet} \tag{1}$$

Where:

h	is the head pressure in psi
P_{oulet}	is the outlet pressure in psi
P_{inlet}	is the inlet pressure in psi

Equation (2) is used to determine efficiency of the pump. It is defined as the ratio between fluid power and rotor power.

$$\eta = \frac{P_h}{P_r} \tag{2}$$

$$P_h = \frac{q\rho g h}{3600} \tag{3}$$

$$P_r = \frac{\pi Q N}{30} \tag{4}$$

Where:

η	is the efficiency
P_h	is the fluid power in W
P_r	is the impeller power in W
q	is the dynamic pressure in psi

ρ	is the density in slug/ft^3
g	is the acceleration due to gravity in ft/s^2
Q	is the impeller torque in lb-ft
N	is the impeller rotation in RPM

To limit variables for optimization, the housing geometry is kept constant and impeller geometry is varied. Only the blade angles and number of blades is analyzed in the sensitivity study, with all other parameters held constant.

Figure 3.1 shows the effect of blade count on the pump performance while maintaining the baseline inflow and outflow blade angles. Note that the baseline is marked with an asterisk. From the figure, it is observed that there is slight performance gain when going from 4 to 5 blades. The analytical model shows that the performance gain hits a plateau with increasing the blade count.

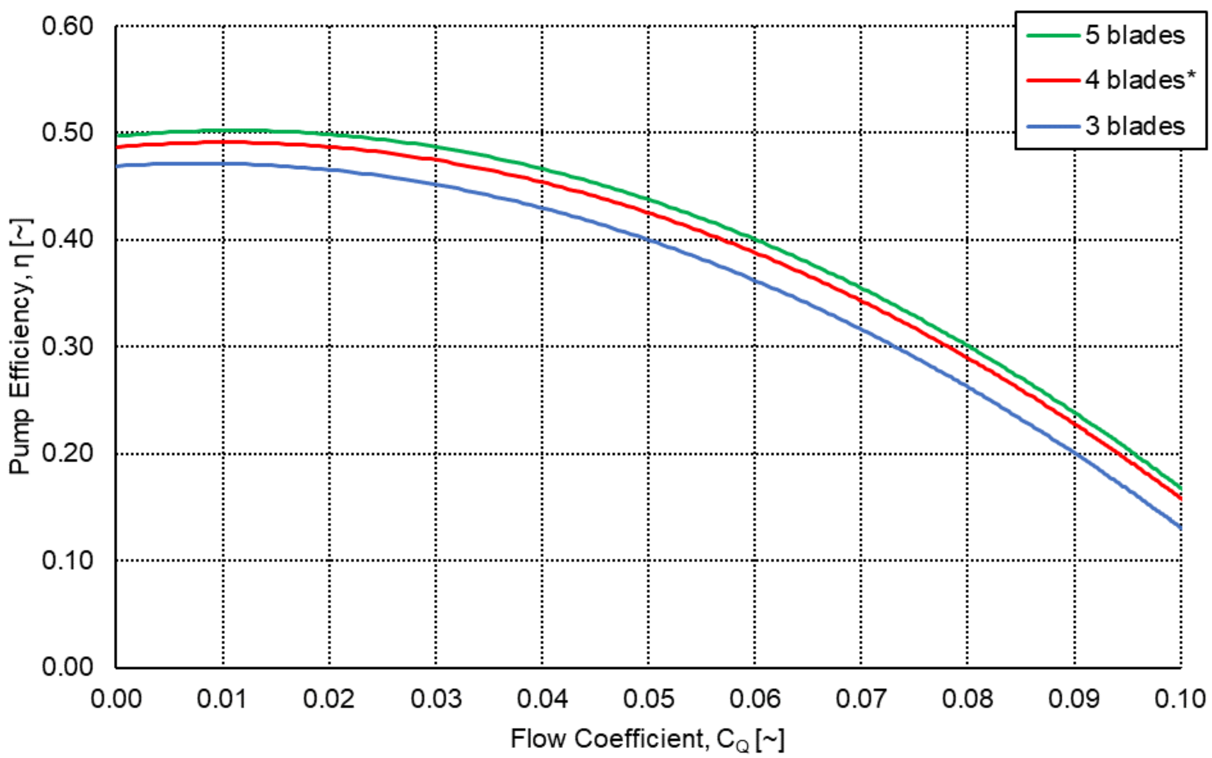

Figure 3.1 Effect of Blade Count on Pump Performance

Flow coefficient is defined as:

$$C_Q = \frac{Q}{(N/60)D_2^3} \tag{5}$$

Where:

C_Q is the flow coefficient

D is the impeller diameter in ft

N is the revolution in rev/s

Q is the flow rate ft^3/s

Figure 3.2 shows the effect of blade exit angle on pump performance while maintaining the baseline blade count. From the figure it can be seen that performance gain is significantly affected by decreasing the blade exit angle.

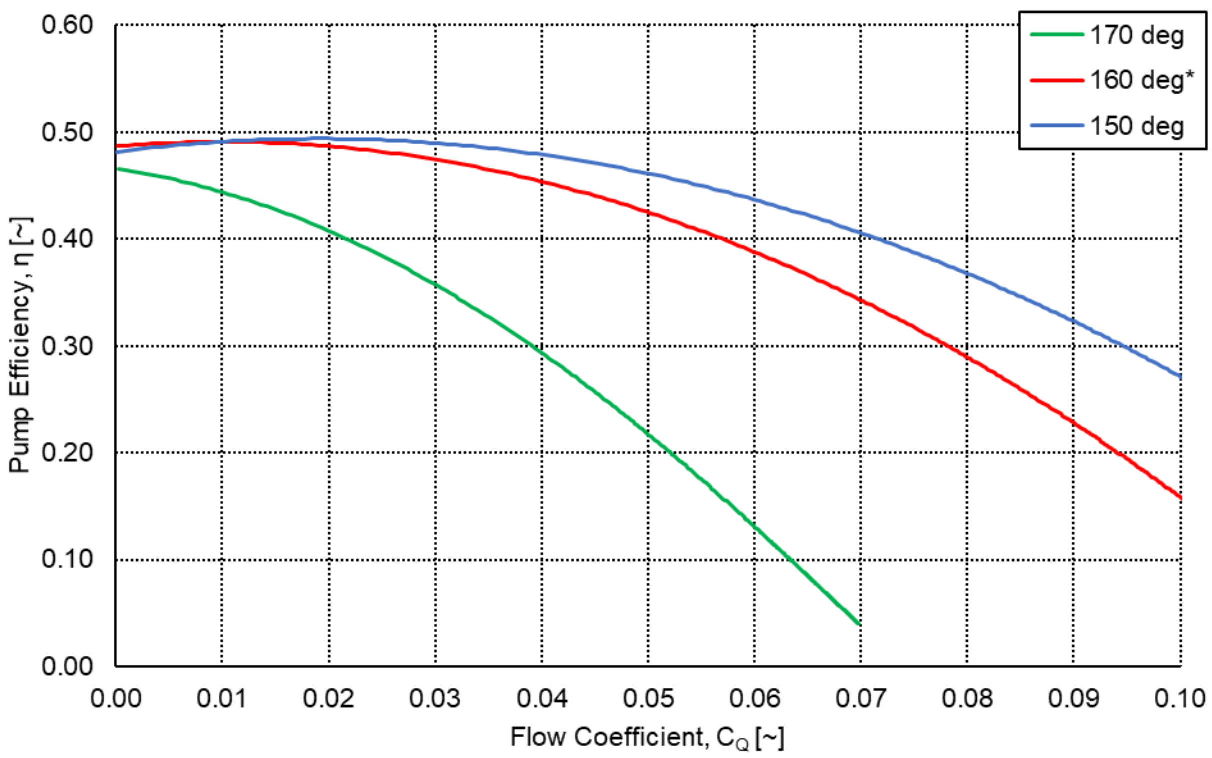

Figure 3.2 Effect of Blade Exit Angle on Pump Performance

Using the information in Figure 3.1 and Figure 3.2 as a guide. The updated pump impeller is modified to have 6 blades with a blade outflow angle of 130 degrees. Table 3.2 shows the pump performance comparison of the baseline and the updated pump design. The analytical model predicts an increase in head pressure of 5.1% without a significant penalty to efficiency.

Table 3.2 Analytical Pump Performance Comparison

60 LPM at 5,000 RPM	Head Pressure [psi]	Efficiency [~]
4bl-115160 Baseline	48.8	0.23
6bl-110130	51.3	0.21

4. Detailed Design

From the sensitivity studies, the most promising design from the preliminary design is run in CFD. STAR-CCM+ (Reference 1) is used to simulate the pump performance. Based on past project experience the physics model and mesh refinements are set to capture the performance of the pump. Table 4.1 shows the pump performance of the updated pump design.

Table 4.1 CFD Pump Performance

60 LPM at 5,000 RPM	Head Pressure [psi]	Efficiency [~]
6bl-110130	53.8	0.34

For this paper, CFD is used to visualize the flow in the updated pump and to verify the head pressure against the analytical model. Optimization of the casing is not considered. Comparing Table 3.2 and Table 4.1, CFD predicts a higher head pressure than the analytical model. This is to be expected since the analytical model does not account for any casing effects. Albeit CFD showing a higher head pressure than the analytical model, the values are in good agreement. Figure 4.1 shows the velocity plot of the improved pump. The figure shows the internal velocity at the impeller (top view) and at the outlet (side view). As observed, there is flow separation and circulation at the tongue and at the impeller exit into the outlet section.

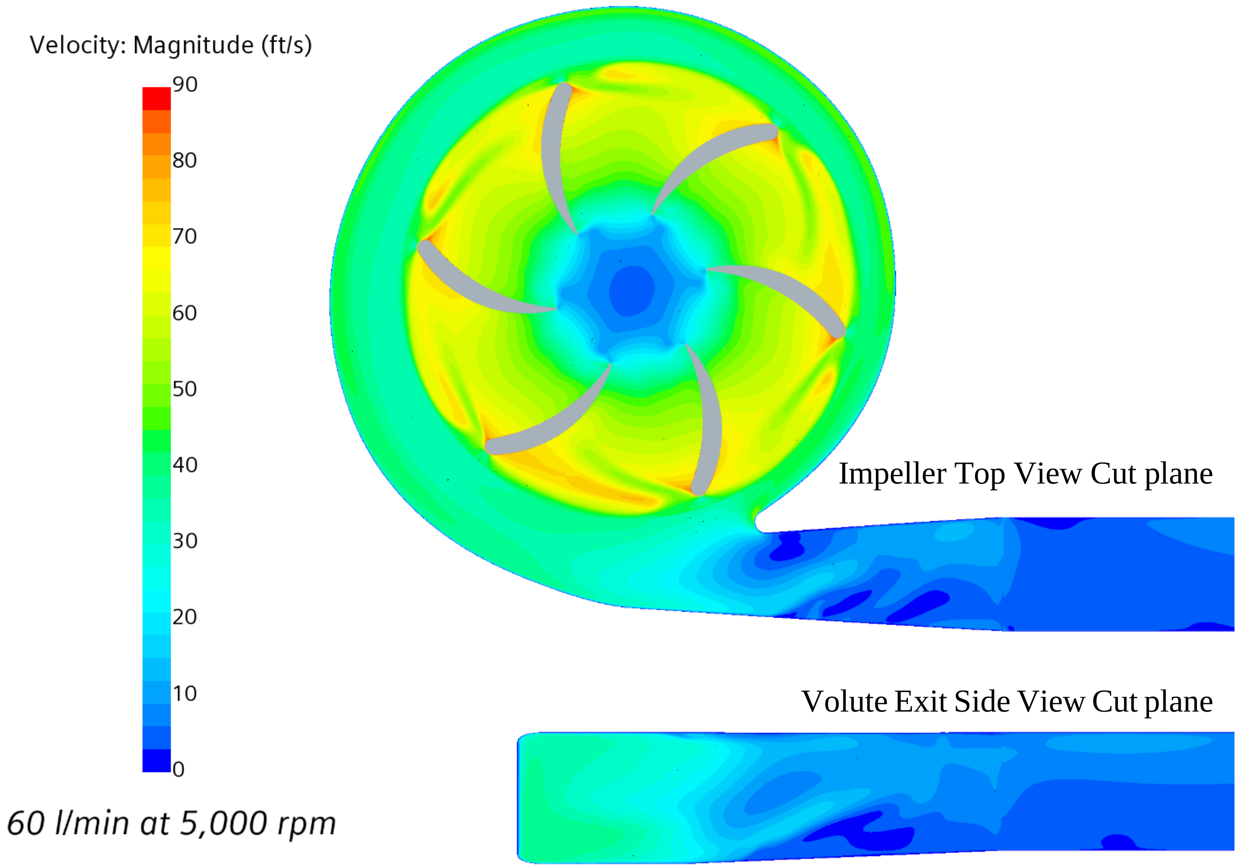

Figure 4.1 Velocity Contour Plot of the Improved Pump

Figure 4.2 shows the velocity around the impeller. Note that the scale is set to show case the flow around the leading edge of the blades.

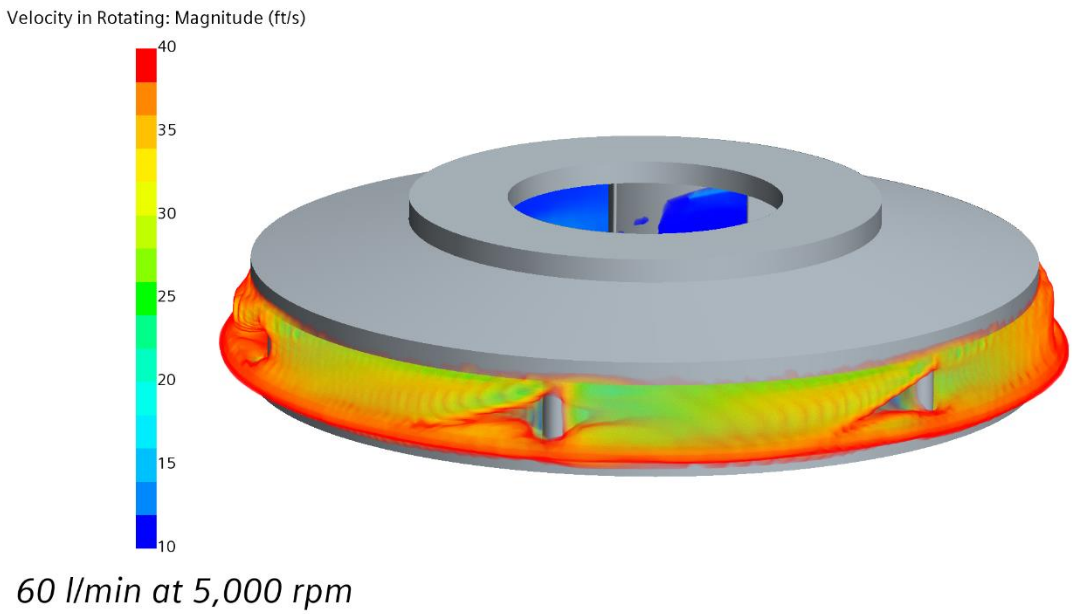

Figure 4.2 Velocity Around the Impeller

Figure 4.3 Shows the velocity streamlines in the updated pump.

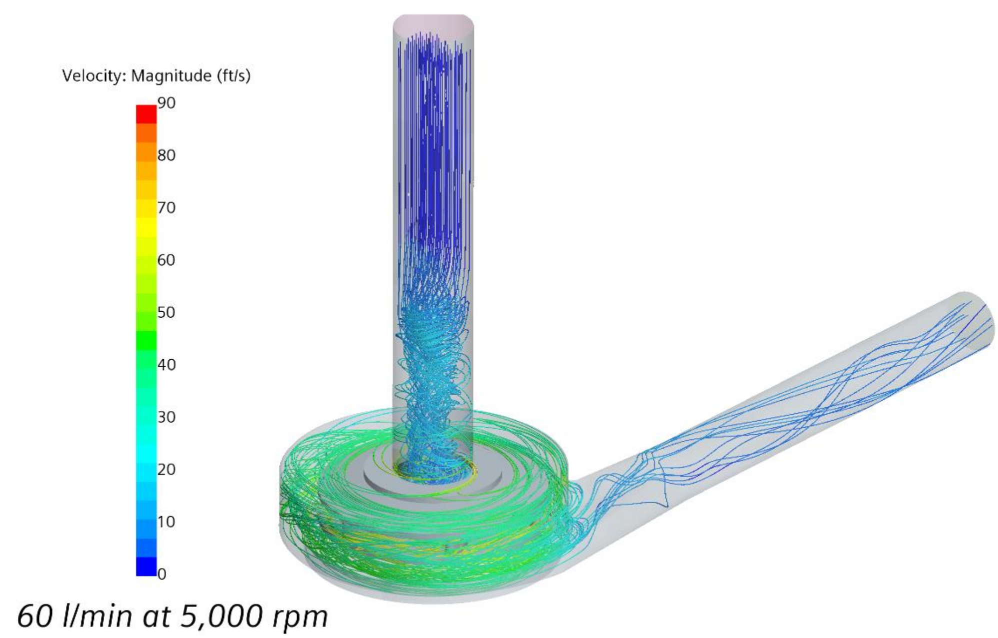

Figure 4.3 Velocity Streamlines in Updated Pump Design

5. Conclusions

Utilizing the in-house analytical tool, various case studies can be rapidly generated based on the desired optimization point. For this paper the optimization is on the head pressure. From the baseline pump to the updated pump, the head pressure is increased by 5.1%. without a significant penalty to efficiency. Additional optimization can be conducted on the casing to reduce regions of recirculation at the volute exit. This will further increase the total head pressure.

6. References

1. Anon., STAR-CCM+ 2020.1.1, build 15.02.007 (win64/intel 18.3-r8 Double Precision), Siemens PLM Software Inc., 2020

7. Further Information

Please contact Design, Analysis and Research Corporation for more information.