A Study on the Change of Displacement according to the Drainage Pump Outlet Shape and Size of Combat Equipment.

Dae Un Kim*, Ki Jung Lee

*1Consolidated Maintenance Depot Republic of Korea Army, St Uichang-daero 783
1 Dong-eup, Uichang-gu Changwon-si Gyeongsangnam-do Republic of Korea.

Email Id: akdong48@nate.com

Abstract. Combat equipment is designed to cross the river on its own. The important thing here is that first, the inflow of water should be minimal, and the second is draining out the inside of the inflow. In this study, we have studied how to effectively discharge the incoming water. The purpose of this study is to compare the flow rate, so the pump pressure was not considered. The flow analysis was carried out only by considering the head pressure and atmospheric pressure. The outlet shape was found to have 13% more emissions than the spiral, which caused vortices due to the helical grooves and protrusions inside the hole, resulting in the loss of flow energy due to the increase in frictional resistance. As a result of checking the drainage according to the size, when the hole size is small, the drainage of water is gradually progressed. When the hole size is large, the drainage is increased rapidly and it is confirmed that it is discharged in a short time. If the hole size is infinite, the effect of drowning is maximized, but it may affect the structural strength of combat equipment, so it should be considered when designing the size of the outlet. Drainage pumps can save the combat equipment from the risk of flooding for water inflows below a certain level. Therefore, when the drainage pump is maintained, the pump efficiency should be guaranteed by thorough inspection and performance test.

Keywords Combat equipment, The crossing of river, Drainage pump, vortex effect, Flooding, Minimum flow inflow

1. Introduction

Korea is a beautiful country with a good combination of mountains and rivers and four seasons. Since the Korean War in 1950 divided into North and South, always prepare for enemy provocations. However, due to the mountains and rivers that make up more than half of the country, there are many restrictions on the maneuverability of combat equipment. Topographical shortcomings must be overcome to ensure maneuverability and combat capability.

Therefore, there are many special cases where you have to cross the river using combat equipment, so always practice through training. Doha is used to cross rivers, in 'Doha', temporary bridges such as Fig. 1 can be used to cross rivers, and even combat equipment can cross rivers. Because the former uses bridges, the risk of immersion of combat equipment in the river is minimal. However, in the latter case, the risk of flooding is increased if a hole in the combat equipment is created or confidentiality is not maintained. In this case, the drainage pump operates to discharge the water introduced into the outside, in case of inflow greater than the discharge, the combat equipment is not safe from the risk of flooding. Therefore, this study investigated the difference of water discharge by the flow analysis by changing the shape of spiral and straight drains at the end of the drainage pump of combat equipment. In addition, the discharge of water per hour was compared and analyzed by varying the size of the outlet diameter.
2. Characteristics of the drainage pump

Drainage pumps used in combat equipment differ in their shape and characteristics, and in different locations. Fig. 2 represents a drain pump that is widely used in combat equipment, and a suction mesh is provided with a mesh of circular mesh to prevent the intake of foreign substances.

Table 1 represents the capacity of the drain pump and it can be confirmed that represents 20Amps phase differences at each head. The maximum capacity of the drain pump is 174 ℓ per minute and the maximum head is 6.3m. In addition, the maximum head of the drain pump used in combat equipment is less than 1.8m, so the maximum pump capacity is 174LPM.
Table 1. Characteristics of the drainage pump

| Hydraulic Head | 1.8m | 3.0m | 4.5m | Max Hydraulic Head (6.3m) |
|----------------|------|------|------|--------------------------|
| LPM, 20Amps   | 174  | 132  | 79   | 0                        |

| Input Voltage | 27.5 VDC         |
|---------------|------------------|
| Pump Weight   | 5kg              |
| Pump Capacity | 174 l/min        |

Figure 3. Performance graph of the pump

3. Preliminary research analysis

In order to secure the reliability of the fluid analysis, similar studies were found in the previous studies. The results of Bae Kang-yeol's research confirm the numerical analysis. To verify the verification, a comparative verification of the numerical analysis technique was performed under the same conditions as the pressure test results in the longitudinal direction in the narrow rectangular channels of 'Brackill' and 'Kandikar'. Applied fluid and fluid temperature were water 26.6°C, which was the same condition as the experiment, and the flow rate boundary condition was applied to the inlet and the pressure boundary condition to the outlet.
The steady state laminar flow analysis was performed. Fig. 3 represents the experimental and analytical model and the pressure measurement position. The experimental results show that the results are in good agreement with the minimum error rate of 0.116% and the maximum error rate of 0.542%. Therefore, the validity of the numerical method used in this study was verified. Fig. 4 is a graph comparing the pressure values for each point in the experiment and numerical analysis.[1]
water to rise inside the combat equipment is shortened, thereby increasing the risk of flooding. Therefore, the shape of the exit hole was assumed to be straight and spiral, and the effective emissions were compared and analyzed. Since the flow analysis is for comparison of the flow rate, only the head pressure is applied without applying the pump pressure. The model is the same as Fig. 6, and the analysis is performed by dividing the straight and spiral. The physical properties of the fluid used for the flow analysis were 25°C at room temperature and a density of 997 kg/m³ was used.

![3D modelling](image)

(a). Straight hole  
(b). Spiral hole

**Figure 6.** 3D modelling

The flow analysis code used for flow analysis was ANSYS CFX 18.0, and the turbulence model used Two-Equation K-ε. In order to derive the calculation result in the steady state of the flow, the steady state equation of the following equation is used.

Continuity Equation

\[
\frac{\partial}{\partial x_i} (\rho U_i) = 0
\]

Momentum Equation

\[
\frac{\partial}{\partial x_i} (\rho U_i U_j) = - \frac{\partial p'}{\partial x_i} - \frac{\partial}{\partial x_i} \left[ \mu_{eff} \left( \frac{\partial U_i}{\partial x_i} + \frac{\partial U_j}{\partial x_j} \right) \right] + S_M
\]

\[p' = p + \frac{2}{3} \rho k + \frac{2}{3} \mu_{eff} \frac{\partial U_k}{\partial x_k},\]

\[\mu_{eff} = \mu + \mu_i,\]

\[\mu_i = C_{\mu} \rho \frac{k^2}{\epsilon},\]
Where SM is the sum of Body Force and Ueff is the turbulent effective viscosity.

Turbulence Kinetic Energy Equation

$$\frac{\partial}{\partial x_i} (\rho U_j U_k) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + P_k - \rho \varepsilon + P_{eb}$$

Turbulence Dissipation Rate Equation

$$\frac{\partial}{\partial x_i} (\rho \varepsilon U_k) = \frac{\partial}{\partial x_i} \left[ \frac{\varepsilon}{\sigma_k} \frac{\partial \varepsilon}{\partial x_i} \right] + \frac{\varepsilon}{k} (c_{et} P_k - C_{\varepsilon-2} \rho \varepsilon + C_{\varepsilon t} P_{eb})$$

$$P_k = \mu_t \left( \frac{\delta U_1}{\delta x_i} \right.^2 \frac{\delta U_1}{\delta x_i} + \frac{2}{3} \frac{\delta U_2}{\delta x_i} \frac{\delta U_2}{\delta x_i} + \rho k.$$ 

Fig. 7 and Fig. 8 represent the streamline according to the velocity distribution in the whole area and the cross section. In the case of the straight type (a), the fluid flows out in a straight line without forming a vortex inside the hole.

![Streamline Distribution](image)

(a) Straight hole  (b) Spiral hole

**Figure 7.** Speed distribution of the whole area

In the spiral of (b), it was found that vortices occurred due to the influence of the spiral groove inside the hole, resulting in a loss of flow energy.
As shown in Table 2, the straight type produced about 13% more emissions than the spiral. When the velocity distribution was inferred, it was found that the vortex occurred due to the influence of the spiral groove and protrusion inside the hole. As a result, it was found that the frictional resistance was increased, resulting in the loss of flow energy.
5. Estimation of discharge flow rate according to outlet diameter change

In this study, as in the fluid analysis of Chapter 4, it is assumed that the pressure acts only on gravity and head because the purpose is to compare flow rates. If natural discharge of the same volume flows downward, we want to predict the discharge and time according to the change of outlet diameter. Numerical three-dimensional modeling is the same as Fig. 9, and the volume of the upper fluid is based on 10ℓ. The discharge diameter was analyzed by flow analysis for two diameter changes of 1 1/4inch and 1/2inch(12.7mm). Since the analytical model is symmetrical as in (a) of Fig. 9, the analysis was conducted by imposing symmetrical conditions using the 1/4 model as shown in (b).

Fig. 10 represents a lattice system for flow analysis, in which Element 2,700,332 NODEs 468,005 were used for the 1/2inch(12.7mm) discharge diameter. A discharge diameter of 1 1/4inch(31.75mm) was used for Element 2,707,996 NODE 469, 383. The working fluid is water at room temperature, and the characteristics of the fluid are the same as Table 3.
Table 3. Characteristics of the fluid

| Property                      | Value       |
|-------------------------------|-------------|
| Density (kg/m³) at 25°C       | 997         |
| Specific Heat Capacity (J/kg.K) | 4181.7     |
| Dynamic Viscosity (kg/m.s)    | 8.899 x 10⁻⁴|
| Thermal Expansivity (/K)      | 2.57 x 10⁻⁴|
| Surface Tension Coefficient (N/m) | 0.072      |

Fig. 11 represents the boundary conditions used in the analysis, and because the 1/4 model was used in the analysis, symmetric boundary conditions were imposed on the plane of symmetry like the red arrow. In addition, the upper part and the lower part are open to the atmosphere, and are assumed to be exposed, and according to physical phenomena, the condition that the outside air is freely in and out is imposed. The gravitational acceleration was applied with a value of 9.81 m/s² in the vertical direction below Fig. 11. The structural analysis tools for the analysis are the same as in Chapter 4. The same equations are applied to the turbulence model and the steady-state equation.

Fig. 12 represents Water isosurface for a discharge diameter of 1/2, and the discharge is almost completed after about 100.5 seconds. It can be seen that the flow rate appears at the beginning of the initial discharge, the flow rate gradually decreases as the discharge proceeds.

At the beginning of the flow field, the head pressure is large because the fluid is filled in the entire area, but when the water level starts to gradually decrease as the fluid starts to discharge, the flow rate decreases gradually because the head pressure decreases.

Fig. 13 shows the equisurface of water in the case where the discharge diameter is 1 1/4. It can be seen...
that the flow rate increased rapidly at the beginning of the initial discharge, and the discharge was almost completed after about 10 seconds.

![Water isosurface of 1/2inch](image1)

**Figure.12.** Water isosurface of 1/2inch

![Water isosurface of 1 1/4inch](image2)

**Figure.13.** Water isosurface of 1 1/4inch

Fig. 14 is a graph showing the change in the volume of the water in the chamber according to the change in diameter according to the diameter. In case of 1/2inch diameter,
The discharge was completed after about 150 seconds, and in case of 1 1/4inch diameter, the discharge was completed after about 40 seconds.

6. Conclusion

When water was introduced into the combat equipment, the discharge by the drain pump was important. Since the performance of the drain pump is determined from the time of production, it is necessary to apply a method of optimizing the shape and size of the outlet to increase the efficiency of the discharge. When the end of the drain pump is designed in a vortex type, it is found that the linear design is justified because loss of flow energy occurs due to the frictional resistance between the spiral groove and the pitch end.

When comparing the drainage pump outlet size, when checking the drainage volume, it was found that the water drained out when the hole size was small. When the hole size was large, the water draining increased sharply at the starting point, it was confirmed that the water discharge is completed in a short time.

However, increasing the outlet diameter in order to increase emissions may adversely affect the structural and mechanical strength of the combat equipment.

References
[1] Bae gang yeol and Jeon jin sung, Study on the hydraulic cylinder flow analysis of Orifice Type, Journal of the korean society for power system engineering., No6, 54-59. 2015.

[2] Cha yeong ho, The proposal of a new drainage system without incline of piping and experiment on drainage flow characteristics., Journal of The society of air conditioning and refrigeration engineering, No5. pp. 452-458. 2005.

[3] Kim kyeong soo, Development of drainage pump for rescue sinking ship, Journal of The korean society of marine engineering, No3, pp248-254. 2015.

[4] ANSYS, Theory Reference for ANSYS.