Design of a Hydrocyclone for Particle Separation in the Seawater Cooling System

Heung June Chung*, Woo Shik Kim
Korea Atomic Energy Research Institute, 111 Daedeok-daero 989Beon-gil, Yuseong-gu, Daejeon, 34057, Korea
**Corresponding author:hjchung@kaeri.re.kr*

1. Introduction

Cyclone is one of the oldest equipment used for the removal or separation of particles in fluids. The main advantages of the use of the cyclones are their simplicity, low costs in construction and maintenance [1]. In recent years, with a rapid computer and computational fluid dynamics(CFD) technique, the use of numerical simulation in predicting the performance of cyclones and designing a new concept become much popular.

In this study, preliminary CFD analyses are carried out to find out the optimal design of hydrocyclone for separation of particles from sea water system. Sea water system can be used as various applications, sea water cooling system and other industrial fields.

Geometric design parameters from CFD analysis is to be used to construction of the pilot scale experimental equipment.

2. Description of the Simulation

In this preliminary CFD analyses, STAR CCM+ is utilized. Figure 1 shows the hydrocyclone geometry in the present study. To model the turbulence flow in a cyclone, K-O model is used. and geometric parameters such as body diameter, cylinder length, inlet width are varied.

CFD approaches are summarized in Table 1 and the generated computational mesh is illustrated in Fig. 2. The CFD analysis conditions are illustrated in detail by Chung [2].

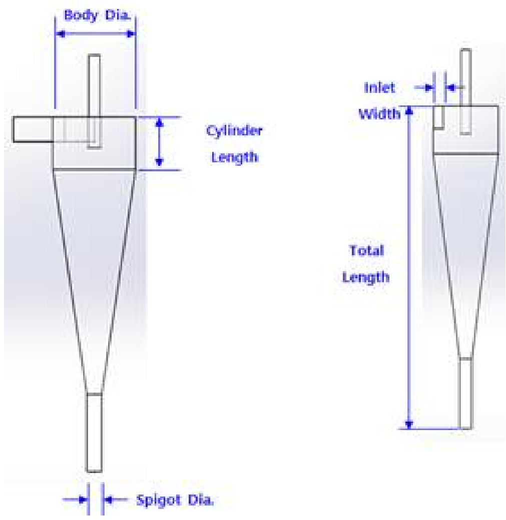

Fig. 1 Geometry of the hydrocyclone

Table 1 Summary of CFD

Mesh (~100,000 cells)	- Trimmed Mesh - Base Size: 5 mm - Maximum Cell Size: 200% - Minimum Surface Size: 10%
Physics	- K-Omega Turbulence - Steady - Gravity - Lagrangian Multiphase
Boundary Conditions	- Liquid (Continuous Phase) ▪ Inlet: Velocity Inlet ▪ Vortex Finder, Spigot: Press. outlet (= 0 Pa) - Solid (Lagrangian Phase) ▪ Injector: Surface Injector (Inlet) / Part Injector (10 x 10 grid) ▪ Diameter: 5 μm ▪ Density: 2000 kg/m^3

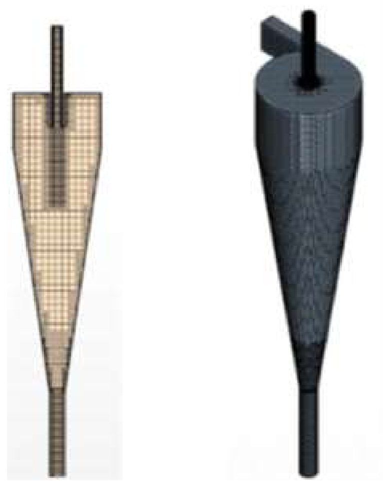
Fig. 2 Illustration of the CFD mesh

3. Results and Conclusions

In this study, to check the geometry effect such as spigot diameter, cylinder length, inlet width and body diameter 9 case study was performed. The aims of the case study is to find out optimal design for desired liquid flow distribution (Spigot/Total=67%), low pressure drop and high particle separation efficiency. The analysis results are shown in Figs. 3-5.

In the case studies, Case2-1[2] show the most proper results according to the present study aims: water distribution of Spigot/Total=67%, inlet pressure of 40kPa and 100% solid separation efficiency.

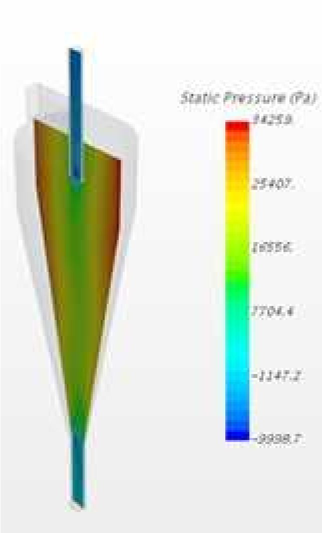

Fig. 3. Pressure distribution in a Cyclone

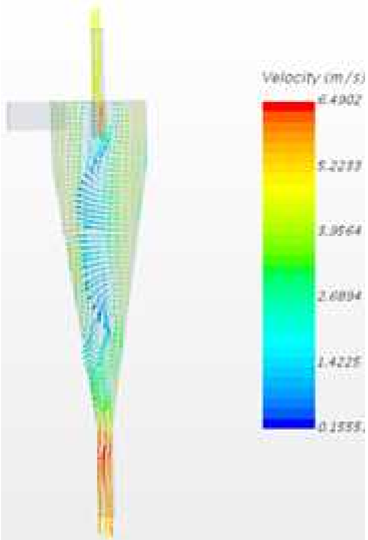

Fig. 4 Liquid velocity distribution in a cyclone

Fig. 5 Particle separation behavior(movie)

REFERENCES

List and number all bibliographical references in 9-point Times, single-spaced, at the end of your paper. When referenced in the text, enclose the citation number in square brackets, for example [1]. It is recommended that the number of references does not exceed five.

[1] A. Seoul, B. Deajeon, and C. Daegu, Article Title, Journal, Vol. 1, No. 1, pp. 1-10, 2002.
[2] D. Busan, Book Title, Publisher, 2000.
pp. 612-613, 1999.