STUDY OF COMPUTATIONAL FLUID DYNAMICS ANALYSIS WITH DESIGN ON THE SPILLWAY OF HYDROELECTRIC POWER PLANT

M. MASUDA
TEPCO Research Institute, Tokyo Electric Power Company Holdings, Inc., Kanagawa, Japan

F. WATANABE
Tepco Systems Corporation, Tokyo, Japan

S. TAKANO
Renewable Power Company, Tokyo Electric Power Company Holdings, Inc., Tokyo, Japan

ABSTRACT

In modification of the designing of the spillway used in our company's hydroelectric power plant, we conducted a hydraulic model experiment based on the conventional method. Using the results of this experiment, we performed a reproducibility analysis with the aim of substituting computational fluid dynamics (CFD) analysis for hydraulic model experiments in further design changes such as cost reduction studies. Based on the results of our detailed examination of the analysis model, such as changing the mesh size or applying an air entrainment model, we concluded the flow of hydraulic model experiment could be reproduced with high accuracy by CFD analysis. As a result, the prospect of substituting the CFD analysis for the experiment was obtained.

1. INTRODUCTION

At our hydroelectric power plant, we are planning to install a new spillway composed of impact-type energy dissipator and stair-type energy dissipator that connect the hydraulic iron pipe and the drainage ditch as shown in Figure 1.

When designing a facility with such a complicated flow, the conventional method uses a hydraulic model experiment to evaluate the influence on the water flow and the facility that cannot be grasped by this calculation against the shape designed by manual calculation. In this case, however, we decided to conduct an analytical study aimed at reducing costs by using computational fluid dynamics (CFD) analysis instead of experimentation.

Specifically, first, a basic hydraulic model experiment was conducted. Next, if the accuracy of the analysis of this experiment can be ensured, it will be judged that CFD analysis can be used as an alternative to the experiment in further design changes such as cost reduction research. Therefore, an analysis method that reproduces the experiment with high accuracy will be considered.

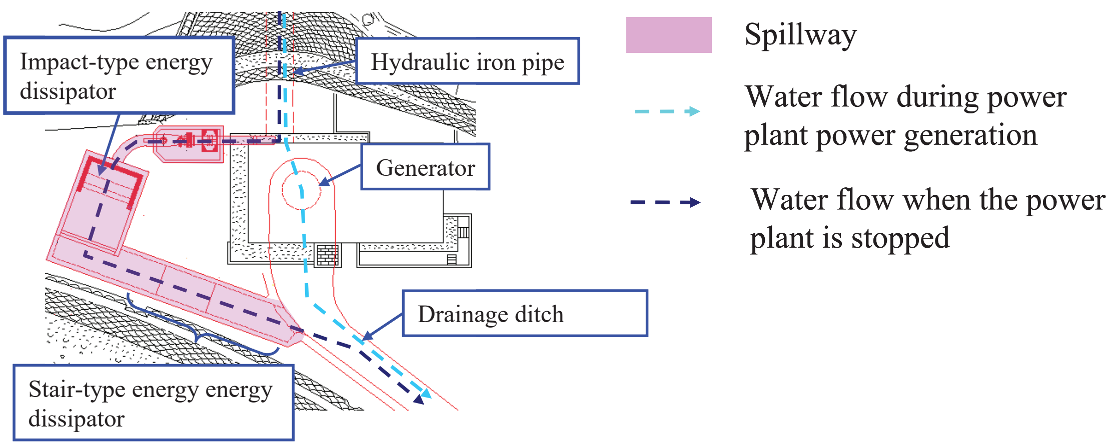


Figure 1 : Overview of the new spillway.

2. HYDRAULIC MODEL EXPERIMENT

The specifications of the hydraulic model experiment are shown in Table 1., the plan view of the hydraulic model experiment equipment and the appearance of the hydraulic model experiment equipment are shown in Figure 2.

In the hydraulic model experiment, the model was created from the discharge pipe separated from the hydraulic iron pipe to the merging section of the drainage ditch, and the scale was created to 1/10 according to the Froude similarity method. Regarding the measurement, the water level at various places in the hydraulic model and the water pressure acting on the impact-type energy dissipator were measured.

Table 1 : The specifications of the hydraulic model experiment.

Scale	1/10
Similarity law	Froude similarity law
Model target	Discharge pipe, Impact-type energy reduction equipment, Stair-type energy reduction equipment, Water discharge way
Contents of measurement	Water levels at various locations and water pressure on the structure

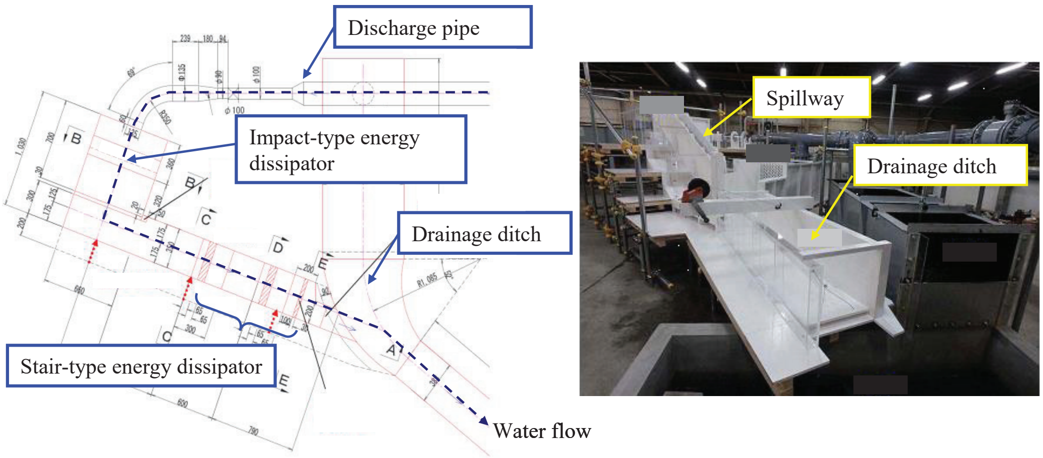


Figure 2 : The plan view of the hydraulic model experiment equipment and the appearance of the hydraulic model experiment equipment.

3. ANALYSIS MODEL

For CFD analysis, we decided to use FLOW-3D, a general-purpose CFD code. An outline of the analysis model is shown in Figure 3.The analysis conditions are as follows.

(1) For shape models, they were modeled with the same range and dimensions as the model experiment.

(2) As for the boundary condition, the upstream is the flow boundary (flow rate is 0.0427m3 / s corresponding to hydraulic model experiment) and the downstream is the pressure boundary (water level designation, water depth is 21.4cm equivalent to hydraulic model experiment).

(3) The surface roughness of the structure was set to 0.2 mm, which is equivalent to a Manning roughness coefficient of 0.01.

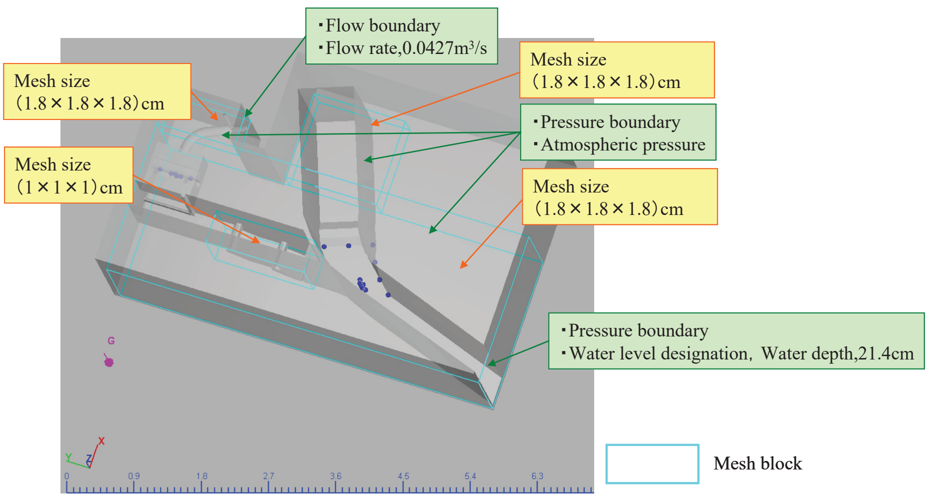


Figure 3 : Overview of analysis model.

Here, when the analytical calculation was performed to reproduce the hydraulic model test results, the following two problems occurred. Therefore, the solution was examined.

(1) There is no flow toward the ceiling in the impact-type energy dissipator.

The flow toward the ceiling generated in the hydraulic model experiment could not be reproduced by the first analysis as shown in Figure 4. About this issue, it was considered necessary to consider the effects of expansion of water due to air entrainment caused by high water turbulence. Therefore, the problem was solved by setting the air entrainment model equipped in the analysis software and adjusting the parameters.

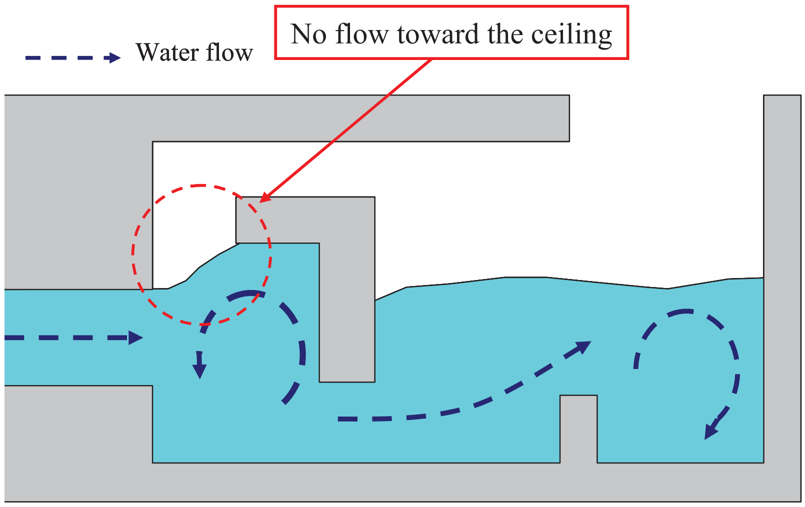


Figure 4 : Conceptual diagram of the flow of the impact-type energy dissipator at the first analysis.

(2) Overflow does not occur in the stair-type energy dissipator.

As shown in Figure 5. , the overflow of the step-type de-energizing facility that occurred in the hydraulic model experiment could not be reproduced in the first analysis. About this issue, since the water depth in the overflow section is shallow, it is considered necessary to subdivide the analysis mesh accordingly. Therefore, a local mesh region was arranged around the stair-type energy dissipator. As a result of the trial, the problem was solved by setting the mesh size to 1×1×1(cm) (the mesh size of the entire analysis region was 1.8×1.8×1.8(cm)).

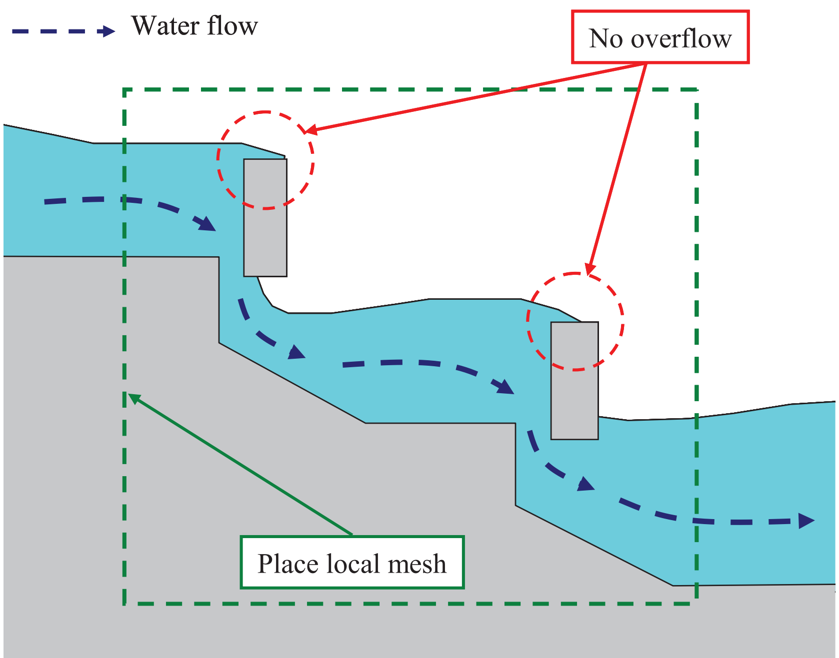


Figure 5 : Conceptual diagram of the flow of the stair-type energy dissipator at the first analysis.

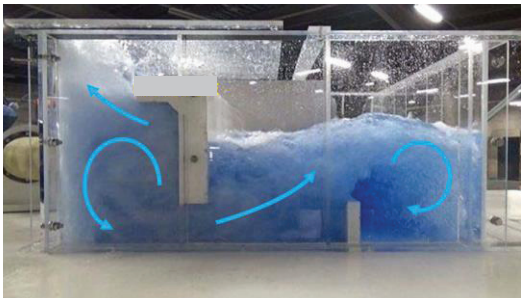
4. ANALYSIS RESULT

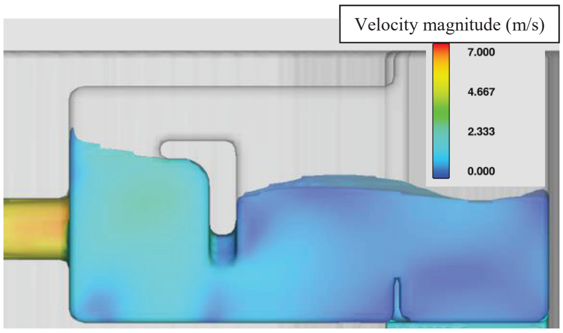
4.1 Comparison of flow conditions in each part

Figure 6. shows the results of a comparison of the CFD analysis results and the hydraulic model experiment results for the flow conditions of the impact-type and stair-type energy dissipator.

The analysis results are considered to be able to reproduce the flow conditions in the hydraulic model experiment.

(Impact-type energy dissipator)

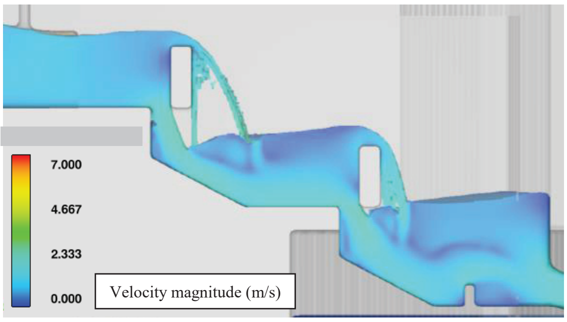




Model experiment flow

Analysis result : Water surface shape and flow velocity distribution

(Stair-type energy dissipator)



Model experiment flow

Analysis result : Water surface shape and flow velocity distribution

Figure 6 : Comparison of flow conditions.

4.2 Comparison of water level and water pressure

First, Figure 7.shows the water level measurement position of the stair-type energy dissipator, and Figure 8.shows the comparison results between the CFD analysis results and the hydraulic model experiment results.

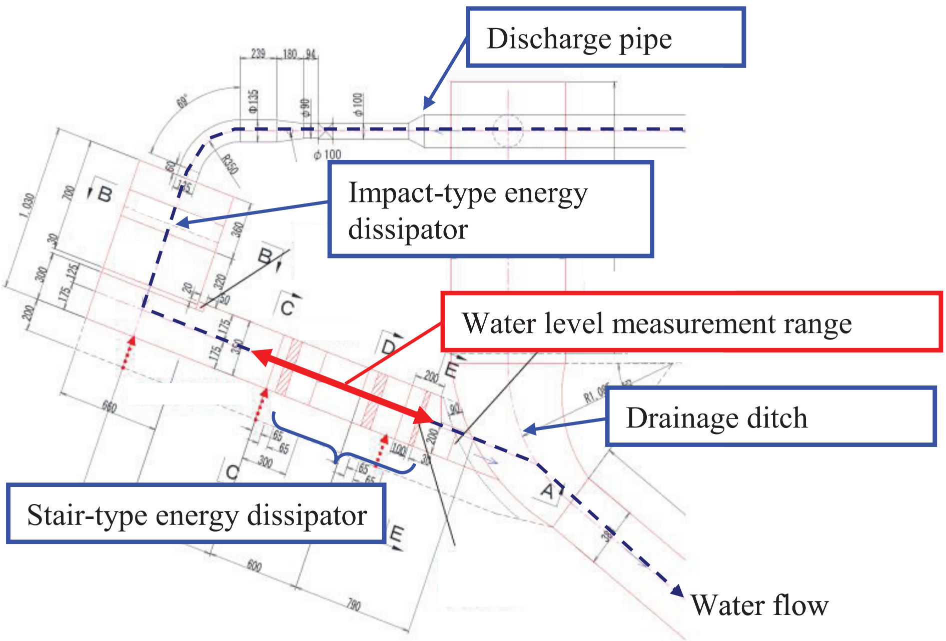


Figure 7 . The water level measurement position of the stair-type energy dissipator .

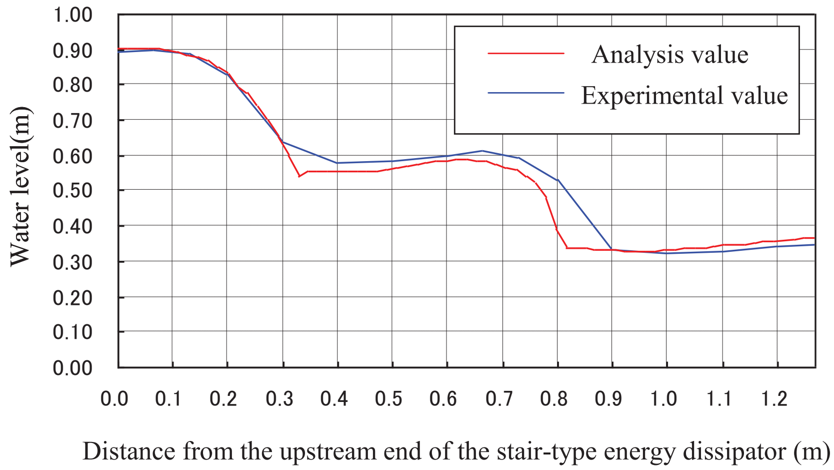


Figure 8 : Results of comparison of centerline water levels in a stair-type energy dissipator .

The water level in the analysis results is lower by about 3cm at the lower stairs than the experimental results, but in other regions, the results were almost consistent with the experimental result.

Next, in impact-type energy dissipator, the water pressure acting on the energy reduction plate against which the inflow water collides is measured. Here, regarding the water pressure acting on the Figure 9 shows the location of the measurement location, and Figure 10 shows the result of comparing the CFD analysis results with the hydraulic model experiment results.

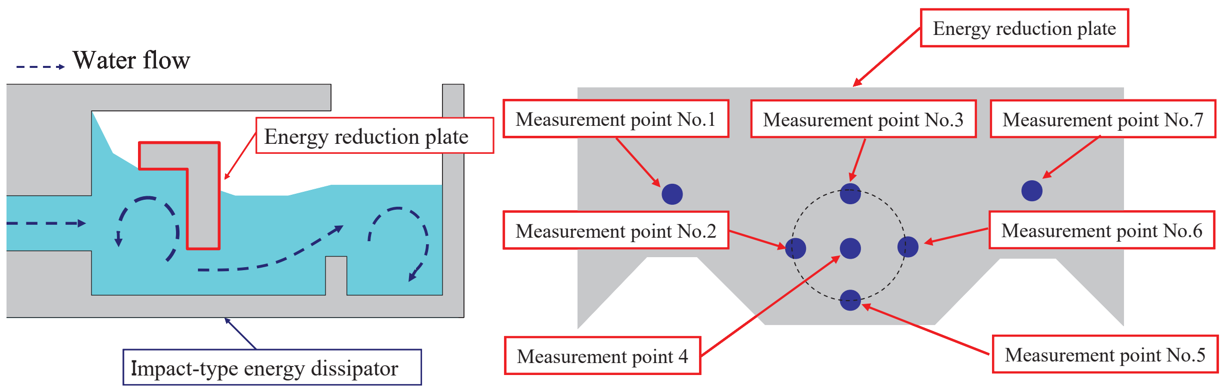


Figure 9 : Water pressure measurement location of the energy reduction plate.

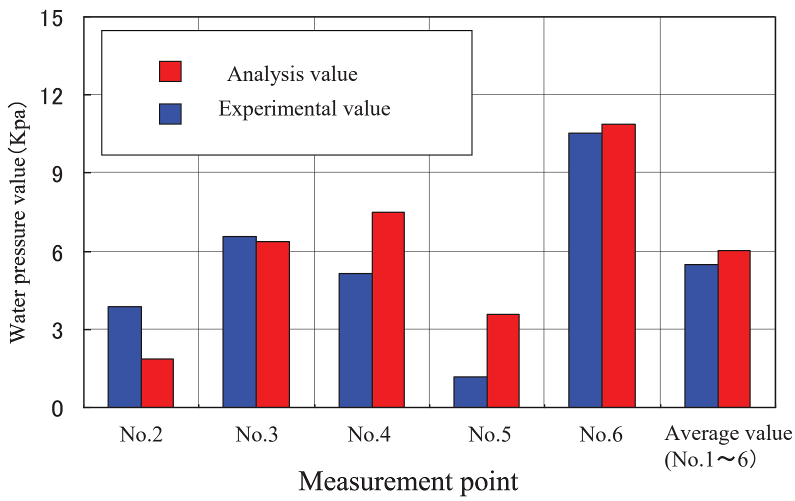


Figure 10 : Comparison results of water pressure acting on the energy reduction plate.

As for the water pressure acting on the impact-type energy dissipator, the analysis results are different at the stations 2, 4, and 5 compared to the model test results. However, the difference from the experimental results with respect to the maximum water pressure and the average value, which are important for design, is very slight, about +0.3 to 0.6kPa, and the analytical results show a tendency similar to the experimental results.

5. CONCLUSIONS

In the design of the spillway of a hydroelectric power plant, a simulation analysis of a basic hy-draulic model experiment was conducted with the aim of replacing CFD analysis with a hydrau-lic model experiment. Through this study, the results of CFD analysis were compared with the results of hydraulic model experiments.

As a result, the CFD analysis results showed the same flow conditions as the hydraulic model experiment, and the validity of the equipment design of the new spillway was confirmed. Over-all, the results were consistent with the hydraulic model experiment, and, the prospect of substi-tuting the CFD analysis for the experiment was obtained.

In the future, we will make further efforts to utilize the analysis results for further design changes such as cost reduction studies and tests using actual machines after construction.