*Sustainable and Safe Dams Around the World – Tournier, Bennett & Bibeau (Eds)*
*© 2019 Canadian Dam Association, ISBN 978-0-367-33422-2*

# Applying CFD analysis to scouring river bed caused by discharge flow from the dam and estimating effectiveness of some countermeasures

K. Hirao, F. Watanabe, S. Ohmori & T. Tsukada
*TEPCO Research Institute, Tokyo Electric Power Company Holdings, Inc., Tokyo, Japan*

T. Kurose
*Renewable Power Company, Tokyo Electric Power Company Holdings, Inc., Tokyo, Japan*

ABSTRACT: Unexpected scour in a river had been observed at the downstream area nearby the discharge channel of the dam-type hydropower station of the Tokyo Electric Power Company Holdings, Inc. We assumed that rotational or three-dimensional (3-D) water flow induced by complicated river bed shape and retaining wall arrangement could be the cause of the scour described above. Applying 3-D computational fluid dynamics (CFD) analysis, we clarify the flow structure relating the scour and study the main factor of the scour. We also validate the effectiveness of some countermeasures against the scour by the CFD analysis. In one countermeasure, we alter the shape of the discharge channel outlet for the purpose of reducing the flow to the observed scour area, and results from the 3-D CFD analysis show the flow accomplished our aim.

RÉSUMÉ: Un affouillement inattendu dans une rivière avait été observé dans la zone en aval à proximité du canal de restitution de la centrale hydroélectrique de type barrage de la Tokyo Electric Power Company Holdings, Inc. Il a été supposé que l'écoulement rotationnel ou tridimensionnel causé par la forme complexe du lit de la rivière et la disposition du mur de soutènement pourrait être la cause de l'affouillement décrit ci-dessus. En utilisant une analyse numérique tridimensionnelle de la dynamique des fluides, la structure de l'écoulement en lien avec l'affouillement a été clarifiée et le facteur principal d'affouillement a été étudié. L'efficacité de certaines contre-mesures contre l'affouillement a été également validée par analyse numérique. Dans une des contre-mesures, la forme de la sortie du canal de restitution a été modifiée afin de réduire le débit vers la zone d'affouillement observée et les résultats de l'analyse tridimensionnelle de la dynamique des fluides montrent que la démarche a atteint l'objectif.

## 1 INTRODUCTION

So far, we consider the safety of our operation and maintenance by the hydraulic model experiment, for example, when we replaced hydropower station facilities with the increase in discharge for maximum power.

On the other hand, with advance of the recent computer and analysis technology, when we investigate the behavior and the phenomenon of the fluid, computational fluid dynamics (CFD) analysis becomes the effective technology as means to study.

From such a background, in late years, we simulate the motion of the water flow by CFD analysis instead of the hydraulic model experiment, and examine for the safety of our operation and maintenance.

In this paper, on the unexpected scour observed at the downstream nearby the discharge channel from our plant, we clarify the flow structure relating the scour and study the main factor of the scour, and then we also validate the effectiveness of some countermeasures against the scour by CFD analysis.

## 2 FLUID BEHAVIOR AROUND THE SCOUR AREA

### 2.1 *Location of the observed scour*

Figure 1 shows two scour areas named A and B. They indicate where we observed at the downstream area, A is the left side, B is downward from the discharge channel.

### 2.2 *Results from the 3-D CFD analysis*

Figure 2 shows the distribution of flow velocity arround the scour area.

Results from CFD show that the scour area A and B is faster than the other area, and the maximum flow velocity is about 9m/s arround the scour area A. In addition, the discharge water flow in the scour area directly.

Figure 3 shows the streamline of the discharge flow.

As a result of CFD analysis, it shows that a part of the discharge water flow rotationaly toward the scour area.

According to the above results, we clarify that rotational or three-dimensional (3-D) water flow induced by complicated riverbed shape and the discharge water flowed directly in the scour area could be the cause of the scour.

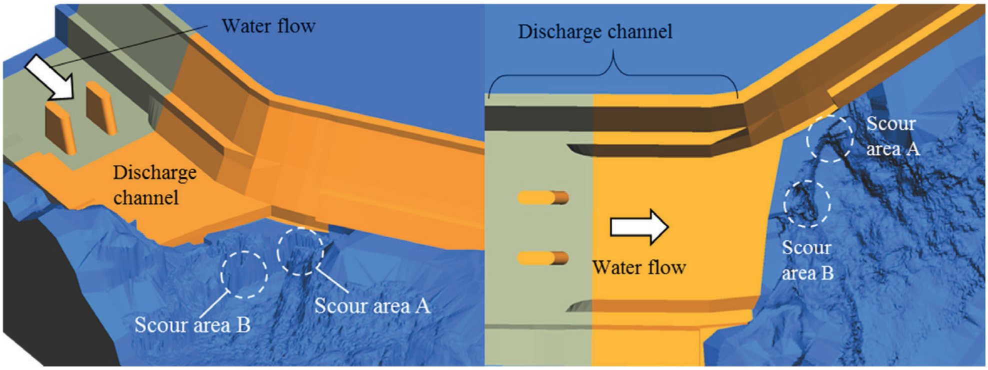


Figure 1. Observed scour area

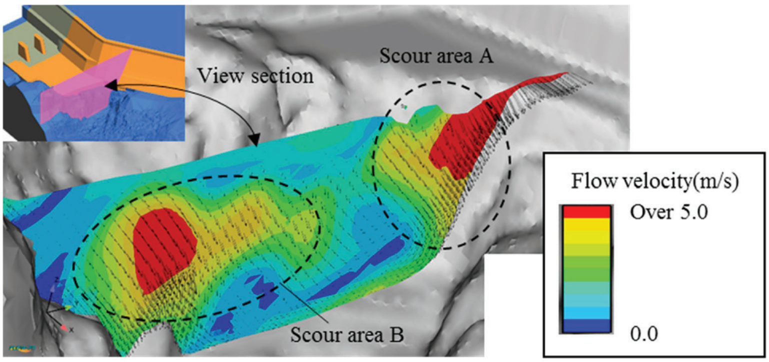


Figure 2. The distribution of flow velocity arround the scour area

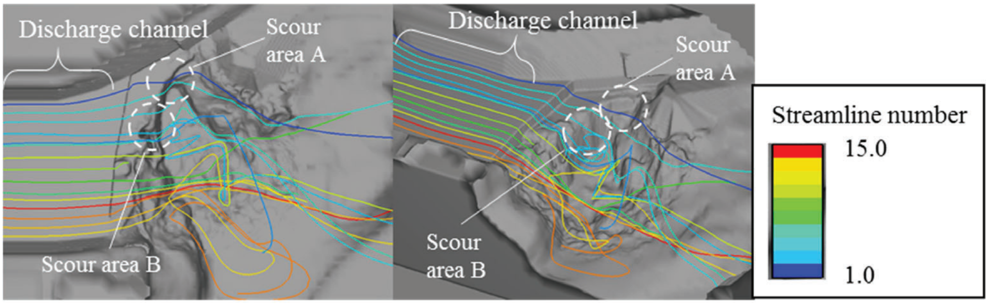


Figure 3. Analysis result described by streamline

## 3 EVALUATION OF THE MEASURES PLAN

### 3.1 *Examination of the measures plan*

Based on the analysis result mentioned above, we focus on the flow to go from the discharge channel to the scour area directly.

At first, for the purpose of reducing this flow, we examine some countermeasures to change the flow direction at the discharge channel edge.

Specially, Figure 4 shows a case setting up the block.

### 3.2 *Evaluation by the analysis*

Figure 5 shows the streamline of the discharge flow on the countermeasure plan (Figure 4).

As a result of analysis, it shows that the almost discharge water don't flow directly toward the scour area, and the main stream can shift to the right side at the discharge channel edge.

On the other hand, it also shows some discharge flow separated from the main stream, and make 'secondary flow' toward the scour area.

Next, Figure 6 shows the distribution of flow velocity arround the scour area in the countermeasure plan (Figure 4).

The above-mentioned secondary flow pass through the scour area B, and upflow along the wall near the scour area. Then, this flow velocity is about 2m/s near the scour area A and slower than the case of no block (4–9m/s). For these results, these countermeasures could

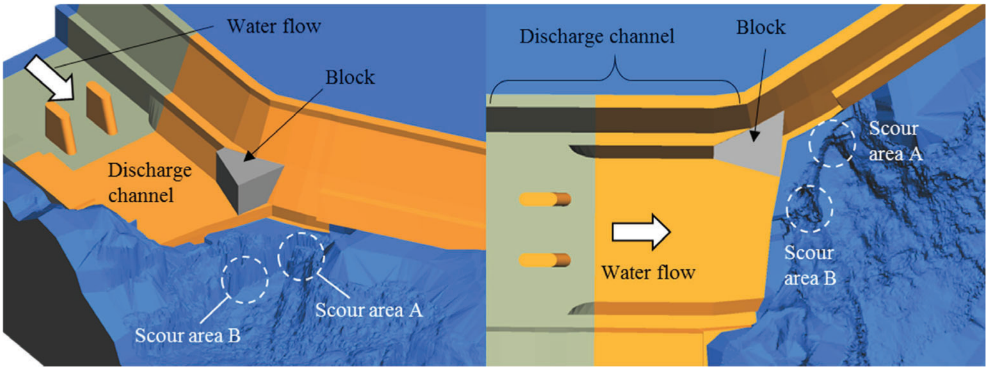


Figure 4. One countermeasure plan

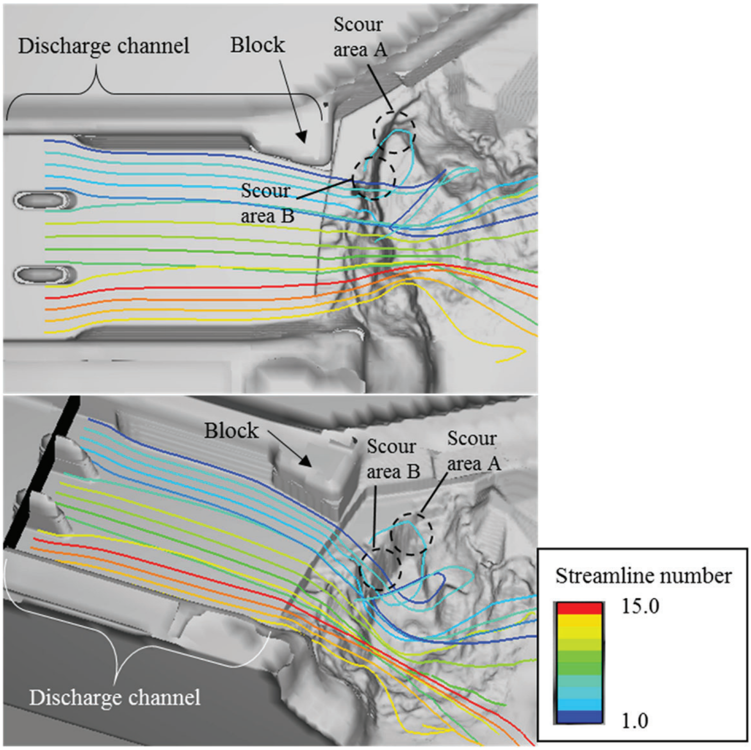


Figure 5. Analysis result described by streamline on the countermeasure plan (Figure 4)

reduce water to flow into the scour area, so we evaluate it could be effective countermeasures against the scour area A.

Last year, we installed a simple block in the discharge channel of our plant, and then we are confirming the effect now. In addition, the flow condition is improved like results of CFD analysis.

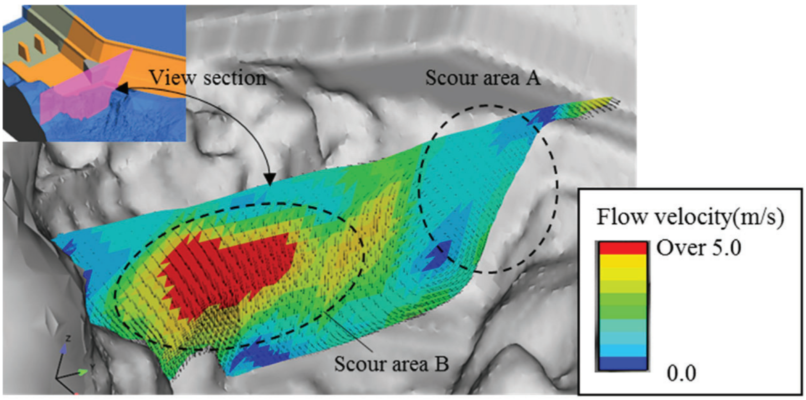


Figure 6. The distribution of flow velocity arround the scour area in the countermeasure plan (Figure 4)

## 4 CONCLUSIONS

We applied the 3-D CFD analysis to clarify the behavior of the fluid nearby the scour area A and B.

The results from CFD analysis show that the discharge water flow in toward the scour area and this flow velocity at the scour area is faster compared with the surrounding area.

In addition, they show that rotational or three-dimensional current is induced by complicated riverbed shape, so we recognize that these factors described above could be the main cause of the scour.

Based on the analysis results mentioned above, we focus on the flow in the direction of the scour area directly and consider some countermeasures.

In these countermeasures, we also validate the effectiveness of preventing the scour by the CFD analysis.

The improved flow near the scour area A is reduced from maximum velocity 9m/s to about 2m/s, so we judge our countermeasures is available against the scour area A.