# Survey/investigation regarding analysis of hydraulic phenomena due to CFD (specific research)

| Year of Research | 2012～2014 | Implementation of anti-inundation measures |
|---|---|---|

**(Purpose)**

Recently, in the long-distance inverted syphon drainage pipe system, accidents such as blowing off of the manhole lid, which is caused by air compression, are reported along with inflow of rainwater to the reservoir pipe. Until now, although the flow in the inverted syphon pipe had been reproduced by means of hydraulic model experiments, when the air was examined, there was a problem due to difficulty in the reappearance of a small model.

In this report, we analyzed the flow in the inverted syphon pipe system using CFD (Computational Fluid Dynamics), and our aim was to obtain the same result as that of a hydraulic model experiment. Moreover, we aimed to verify the availability of the calculation technique of "a two-phase algorithm based on the Volume Of Fluid (VOF) method to air-water interface (the VOF method)".

**(Conclusion)**

1) Method

The geometry of this model calculated this time was analogous to the past experiment model, and that model is the inverted syphon pipe system. The calculation model is shown in **Figure 1**, which has a drop-shaft system under the inflow condition. We applied the VOF method for the computational model of the free surface in order to restage the phenomenon of the air mixture. Moreover, we used the zero-equation model for the turbulent flow model.

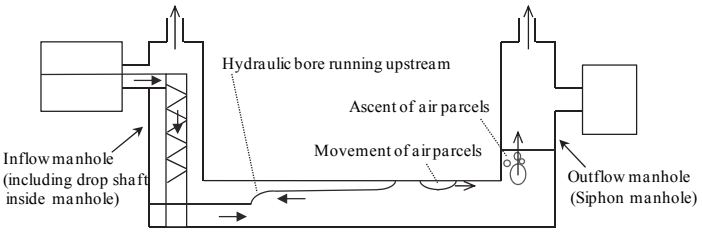


**Figure 1: Phenomenon of inverted syphon in experiment (illustration)**

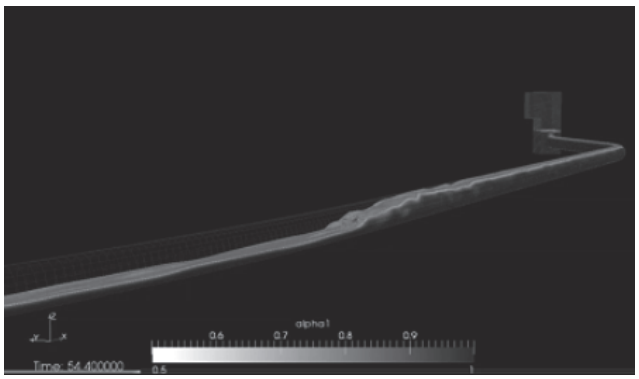


**Figure 2: Bore rushing upstream (CFD analysis results)**

2) Results

After the stream reached the outflow manhole, bore wave phenomenon was observed (see **Figure 2**). The water surface of the bore wave gradually rose, then the pipe was filled by water and the water level of the outflow manhole increased. To the experiment-with-a-model result of having generated the water level in the outflow people hole at the time of a bore generating in the height based on pipes, I had generated in the high position exceeding the top of the pipe, and the analysis result was different. However, comparatively good approximation was obtained about the form (situation of return of a bore wave and the water surface in a pipeline) of the water surface. In particular, between CFD analysis and the past experiment model, the coincidence of the crash of front part of the bore wave was obtained as compared with the experiment with a model.

After a sloshing wave was generated, the air of the pipe top was pushed out by the sloshing wave to the upper stream side, inverse air flow occurred and reached the inflow manhole, and air blew off from the inflow manhole. After the bore wave reached the inflow manhole, the amount of air blow represented the maximum value and the mass of air was scattered. The amount of air discharge at this time is mostly in agreement with the amount of the maximum discharge of about 20 $m^3/s$, and this value was a fitted experimental value.

We checked that the CFD analysis was shown to be beneficial for the analysis of the air-water interface phenomenon and the bore wave.

**(Future prospects)**

From the analysis result of the two-phase algorithm (air-water), we are analyzing the phenomena of the multiphase flow of the air-water in the inverted syphon pipe system. In the future, we would like to analyze much more phenomena. We will utilize these researches and contribute to the construction of a safer sewer system that includes complex phenomena such as air-water mixture.

※ Japan Institute of Wastewater Engineering and Technology
Inquiries ; 2nd Research Department: Hiroshi Kouchiwa, Makoto Ishikawa, Shintaro Sugi [03-5228-6598]

| Key words | Computational Fluid Dynamics (CFD), drop shaft, gas-liquid two-phase flow, bore, inverted syphon, VOF method |
|---|---|