CFD Analysis of the Safety Injection Tank and Fluidic Device

Jai Oan Cho [a], Yacine Addad [b], Yohanes Setiawan Nietiadi [a], Jeong Ik Lee [a*]
[a]*Department of Nuclear and Quantum Engineering, Korea Advanced Institute of Science and Technology (KAIST)*
[b]*Department of Nuclear Engineering, Khalifa University of Science, Technology & Research (KUSTAR)*
[*]*Corresponding author: jeongiklee@kaist.ac.kr*

1. Introduction

The APR 1400 is a large pressurized water reactor (PWR). Just like many other water reactors, it has an emergency core cooling system (ECCS). One of the most important components in the ECCS is the safety injection tank (SIT). Inside the SIT, a fluidic device is installed, which passively controls the mass flow of the safety injection and eliminates the need for low pressure safety injection pumps. As more passive safety mechanisms are being pursued, it has become more important to understand flow structure and the loss mechanism within the fluidic device. Current computational fluid dynamics (CFD) calculations have had limited success in predicting the fluid flow accurately. This study proposes to find a more exact result using CFD and more realistic modeling to predict the performance during accident scenarios more accurately. Additionally, the nitrogen entrainment during SIT flow discharge is a hot issue among regulatory bodies. This research will provide a basis for predicting nitrogen discharge into the core.

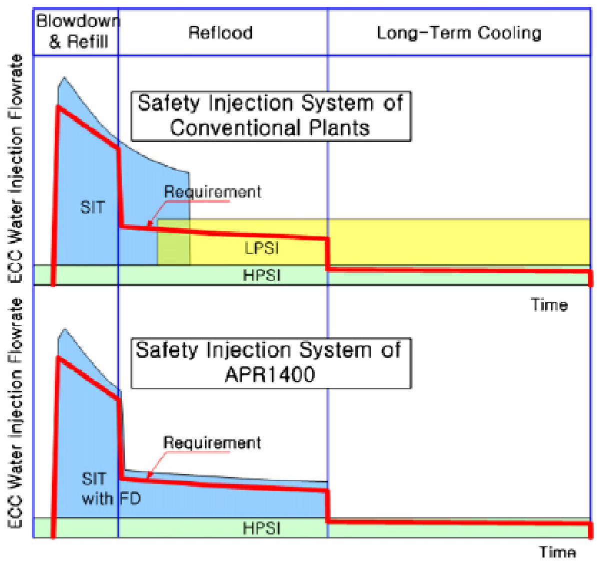

Fig. 1. Mass flow of SITs with and without fluidic devices [1]

2. Preliminary Analysis

CFD calculation is required in order to acquire the loss coefficient. However, numerically the problem is challenging since compressible fluid and incompressible fluid exist in the same large physical problem domain. Because of its complexity, the problem was simplified in many ways to get results in a reasonable amount of time. However, the case was simulated without much simplification in the proposed calculation.

Table I: Research Uniqueness

Previous Works	This Work
Absence of nitrogen	Presence of nitrogen
Geometry simplification	Full geometry in full scale
K-factor applied as calculated value	K-factor not given artificially
Problem domain confined to Fluidic Device	Full tank ı Fluidic Device
Steady State	Transient

A preliminary calculation was run using a coarse grid system. Over 60,000 polyhedral meshes was used with a base size of 20cm. Due to the violent vortex, finer meshes with a base size of 2cm was needed in the fluidic device. The K-epsilon model was used for the computation. The calculation was run under multiphase condition using the VOF model to simulate the nitrogen behavior. The tank was given a constant thermal resistance and constant ambient temperature with convective boundary condition on the tank wall. Lastly, a pressure boundary of 1 bar was given at the end of the discharge pipe.

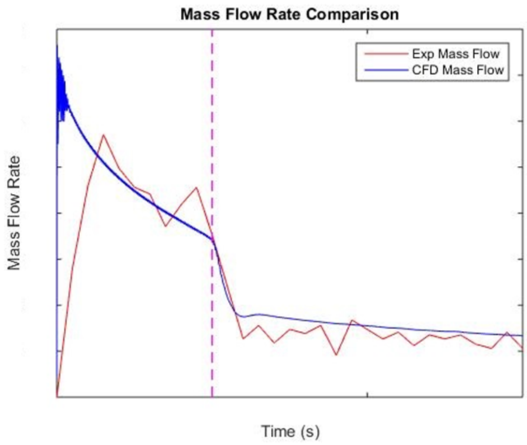

Fig. 2. Mass Flow Rate Comparison of Experiment and CFD

The preliminary calculation matched the experimental results very well even with low resolution grids. The mass flow rate fits the results from the experiment properly and the flow transition from high flow to low flow was simulated smoothly. The 3D visualization of streamlines also shows the vortex in the fluidic device properly. It is noted that the time scale and y-axis values are intentionally deleted to protect the proprietary data.

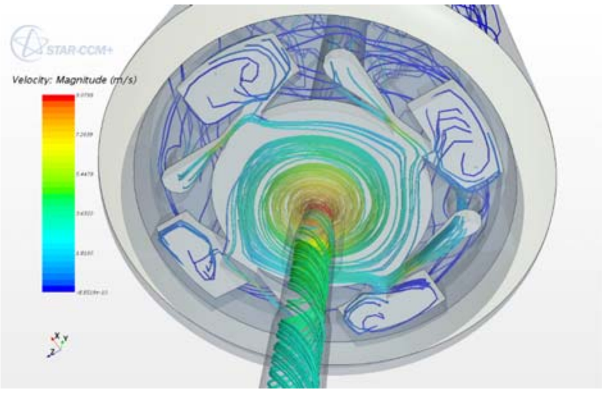

Fig. 3. CFD results of streamline

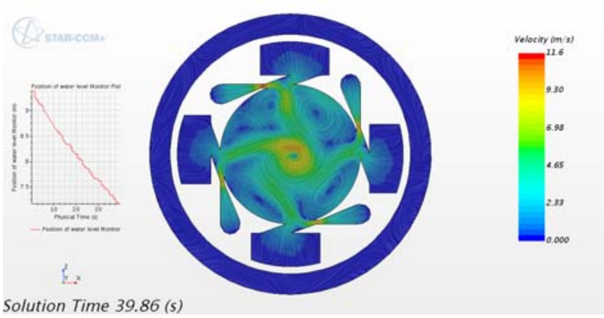

Fig.4. Velocity profile in fluidic device

3. Comprehensive Analysis

After the successful preliminary analysis, a more comprehensive analysis using star ccm+ was proposed. Most configurations were maintained but higher mesh resolution and shorter time step were adopted. The tank itself does not require fine grid but to capture the flow inside the fluidic device, even finer meshes were needed. The mesh base size is 10cm in most parts of the tank. The mesh in the fluidic device has a base size 10% relative to the tank and 2.5% in certain areas.

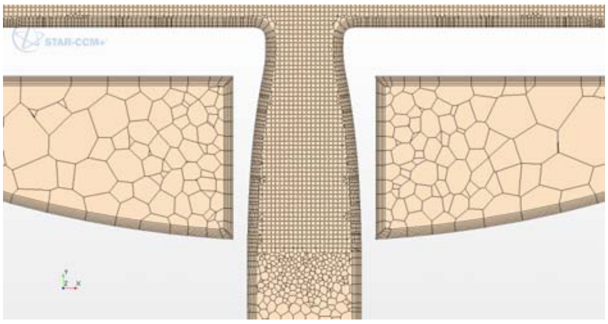

Fig.5. Grid system in discharge pipe

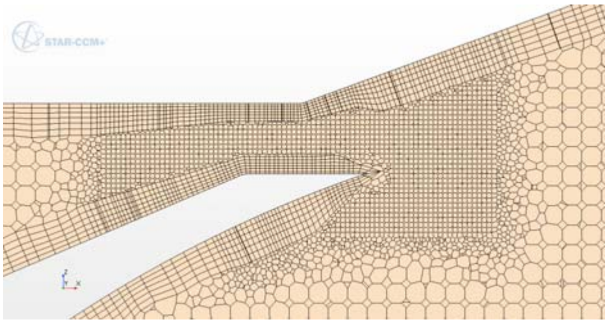

Fig.6. Grid system in fluidic device

6 prism layers are used for the walls with a stretching factor of 1.08. The growth factor is given as 0.1 to reflect the gradual increase of mesh size in the boundary of volume control blocks. Once again, realizable k epsilon model was taken as the turbulence model. Eulerian multiphase model with volume of fluid was used. Implicit unsteady calculation is applied with a big time step in the start until flow stabilizes and then it is changed into 5.0 E-6. Detailed results will be presented in the conference.

4. Conclusions and Future Works

The safety injection tank with fluidic device was analyzed thoroughly using CFD. The preliminary calculation used 60,000 meshes for the initial test calculation. The results fit the experimental results surprisingly despite its coarse grid. Nonetheless, the mesh resolution was increased to capture the vortex in the fluidic device precisely.

Once a detailed CFD computation is finished, a small-scale experiment will be conducted for the given conditions. Using the experimental results and the CFD model, physical models can be improved to fit the results more accurately. The data from CFD and experiments will provide a more accurate k-factor which can later be applied in system code inputs using trips and multiple valves. Finally, the fluidic device design can be optimized for the future SIT designs.

ACKNOWLEDGMENTS

This research was supported by the KUSTAR-KAIST Institute, KAIST, Korea

REFERENCES

[1] I. C. CHU et al., Development of passive flow controlling safety injection tank for APR1400, Nuclear Engineering and Design Vol.238, p.200-206, 2008.