CFD Analysis for Advanced Accumulator

Non-Proprietary Version

December 2009

Revision History

Revision	Page	Description
0	All	Original issued

Mitsubishi Heavy Industries, Ltd.
16-5, Konan 2-chome, Minato-ku
Tokyo 108-8215 Japan

Abstract

The Advanced Accumulator (ACC) developed by MHI has a function of flow switching to change flow rate automatically from large flow rate to small flow rate as the function for the requirement on a Loss of Coolant Accident (LOCA) event. The function is achieved by the flow damper in the accumulator tank. For the purpose of understanding flow characteristics and verification of performance for the ACC, “Full-Height 1/2 Scale Confirmation Test“, which has full heights of the test tank and the standpipe, had been conducted (Ref.1). In large Reynolds number flow, little influence of viscous effects is anticipated, and measured data indicates the same flow characteristics between 1/5 and 1/2 scaled model tests (Ref.1). From the above mentioned reasons, the full scaled ACC with larger Reynolds number is reasonable to assume having the same characteristics as the 1/2-scaled model. Therefore, the characteristics can be used to evaluate full scale ACC.

However, since there is little detailed information measured in the vortex chamber of the flow damper and near throat section in existing experiments, that may make it difficult to show enough evidence about the validity of the extrapolation to full scale. To provide an adequate explanation, a good understanding about the flow dynamics of the subject is indispensable, especially flow inside the vortex chamber and the continuing piping (such as throat, diffuser and injection piping), which may include cavitation phenomena.

Regarding the background mentioned above, a Computational Fluid Dynamics (CFD) analysis was applied to comprehend the flow pattern and the cavitation phenomena in the ACC, and to assist the explanation of the validness for the extrapolation from 1/2 to 1/1 scaled model.
In this investigation, steady-state CFD analysis with cavitation model option was conducted by using 1/2 scaled model and 1/1 scaled model to show the similarity of flow pattern and flow characteristic performance regarding flow rate coefficient between 1/2 and 1/1 scaled size.

Below is the summary and conclusions about this report regarding the CFD analysis.

- The CFD prediction is feasible for large and small characteristic evaluations. In 1/2 scaled model analysis, the correlation between flow rate coefficient (C_v) and cavitation factor (σ_v) reasonably matched with the measured data.
- The validity of the current evaluation approach (Ref.1, Ref.2) for the ACC performance, i.e. the extrapolation from scaled model experiment and the characteristic equations, is uncontradicted regarding scale effect due to comparison of the calculated data in 1/2 scaled model and 1/1 scaled one.

Table of Contents

List of Tables

List of Figures

List of Acronyms

ACC	Advanced Accumulator
APWR	Advanced Pressurized-Water Reactor
CFD	Computational Fluid Dynamics
COL	Combined License
C_v	Flow Rate Coefficient
ECCS	Emergency Core Cooling System
LOCA	Loss of Coolant Accident
MHI	Mitsubishi Heavy Industries, Ltd
RCS	Reactor Coolant System
RSM	Reynolds Stress Model
US	United States of America
σ_v	Cavitation Factor

1.0 INTRODUCTION

This report describes the Mitsubishi Heavy Industries, Ltd. (MHI) Advanced Accumulator (ACC) Computational Fluid Dynamics (CFD) analysis results. The purpose of this document is to describe that the same phenomena of the ACC would occur between test facility and actual scale tank. Review of this Technical Report should increase the efficiency of the US-APWR Design Certification process and any subsequent Combined Licenses (COL) which references the US-APWR Design.

The ACC is an accumulator tank with the flow damper that is partially filled with borated water and is pressurized with nitrogen. It is attached to the primary system with a series of check valves and an isolation valve and is aligned during operation to allow flow into the primary coolant system if the primary system pressure drops below the pressure of the accumulator. The ACC design combines the known advantages and extensive operating experience of a conventional accumulator used for Loss of Coolant Accident (LOCA) mitigation in pressurized water reactors with the inherent reliability of a passive fluidic device to achieve a desired reactor coolant injection flow profile without the need of any moving parts. Incorporation of the ACC into the US-APWR design and LOCA mitigation strategy simplifies a critically important safety system by integrating an inherently reliable passive safety component into the conventional Emergency Core Cooling System (ECCS). This design improvement will allow the elimination of low head safety injection pumps, and increase the amount of time available for the installed backup emergency power system to actuate. It is expected that the use of ACCs rather than low head safety injection pumps in the US-APWR design will reduce the net maintenance and testing workload at nuclear facilities while maintaining a very high level of safety.

Topical Report “The Advanced Accumulator”, MUAP-07001 Revision 2 (Ref.1), has been submitted to describe the principles of operation of the ACC, the important design features, and the extensive analysis and confirmatory testing program conducted. This Technical Report describes CFD analysis results to support discussion of scalability in the Topical Report.

2.0 OBJECTIVE

2.1 Background

Currently, for the purpose of understanding flow characteristics and verification of performance for the ACC , scale model tests excepting the heights of the test tank and the standpipe had been conducted (Ref.1). In large Reynolds number flow, little influence of viscous effects is anticipated, and measured data indicates the same flow characteristics between 1/5 and 1/2 scaled model tests (Ref.1). From the above mentioned reasons, the full scaled ACC with larger Reynolds number is reasonable to assume having the same characteristics as the 1/2-scaled model.
However, since there is little detailed information measured in the vortex chamber of the flow damper and near throat section in existing experiments, that may make it difficult to show enough evidence about the validity of the extrapolation to full scale. To provide an adequate explanation, a good understanding about the flow dynamics of the subject is indispensable, especially flow inside the vortex chamber and the continuing piping (such as throat, diffuser and injection piping), which may include cavitation phenomena.

2.2 Objective

With regards to the background mentioned above, a CFD analysis was applied to comprehend the flow pattern and the cavitation phenomena in the ACC, and to assist the explanation of the validness for the extrapolation from 1/2 to 1/1 scaled model.

In this investigation, steady-state CFD analysis with cavitation model option is conducted, using the 1/2 and 1/1 scaled model. The calculated data is evaluated as the following.

- To evaluate the flow behavior at quasi-steady-states for both small and large flow rate conditions.
- To evaluate the correlation between relevant parameters and its corresponding measured data, in 1/2 scaled model calculations.

3.0 CFD ANALYSIS

3.1 System Descriptions

Accumulator system is one of the subsystem of the ECCS. There are four accumulators, one for each reactor coolant cold leg. The accumulators are vertically mounted cylindrical tanks located outside each steam generator/reactor coolant pump cubicle. The accumulators are passive devices. The accumulators are filled with boric acid water and charged with nitrogen. The accumulators discharge water into the reactor cold leg when the cold leg pressure falls below the accumulator pressure.

The accumulators incorporate internal passive flow dampers, which function to inject a large flow to refill the reactor vessel in the first stage of injection, and then reduce the flow as the accumulator water level drops. When the water level is above the top of the standpipe, water enters the flow damper through both inlets at the top of the standpipe and at the side of the flow damper, and injects water with a large flow rate. When the water level drops below the top of the standpipe, the water enters the flow damper only through the side inlet, and injects water with a relatively low flow rate.

The two series check valves in the supply line to the reactor cold leg are held closed by the pressure differential between the Reactor Coolant System (RCS) and the accumulator charge pressure (approximately 1,600 pounds per square inch differential (psid)). The accumulator water level, boron concentration, and nitrogen charged pressure can all be remotely adjusted during power operations. The accumulators are non-insulated and assume thermal equilibrium with the containment normal operating temperature (approximately 70 to 120°F).

The accumulators are charged nitrogen gas by a flow control valve in a common nitrogen supply line. The failure of the flow control valve is accommodated by a safety valve set at 700 psig and having a (nitrogen) flow capacity of 90,000 ft^3 per hour. Likewise, each accumulator is equipped with a safety valve set at 700 psig and (nitrogen) flow capacity of 90,000 ft^3 per hour, which provides a margin from the normal operating pressure (640 psig), yet precludes overcharging by the associated safety injection pump.

3.2 Analysis Models

Two analysis models were made corresponding to the different flow injections, i.e. large flow injection and small flow injection. This is intended to reduce the number of mesh elements and calculation load in small flow rate condition, where the calculation region is smaller than the one in the large flow rate condition.

3.2.1 Geometrical Modeling

Figures 3.2-1 and 3.2-2 show analysis models for each flow injection case.

(1) For large flow injection

1. The model consists of ACC tank (its height is extended to the water level), a standpipe (including an anti-vortex cap), an anti-vortex plate, a vortex chamber, an outlet nozzle and an injection pipe (to the pressure measurement point), which activate when water level is at large injection. **(See note1)**
2. The inner configuration of the flow damper is precisely modeled.
3. The thickness of casing is neglected, such as vortex chamber casing, ducting casing etc.

(note1) Water level is set at stationary for steady state analysis.

(2) For small flow injection

1. The model mainly consists of a lower part of the standpipe, an anti-vortex plate, a vortex chamber, an outlet nozzle and an injection pipe (to the pressure measurement point), which activate when water level is at small injection. **(See note2)**
2. The inner configuration of the flow damper is precisely modeled.
3. The thickness of casing is neglected, such as vortex chamber casing, ducting casing etc.

(note2) The boundary conditions give flow rate at the inlet of the small flow pipe and pressure at the exit of the injection pipe.

(3) Scaling

1. Two types of scaled model were constructed, 1/2 scaled model and 1/1 model, for the evaluation of scaling effects.
2. 1/1 model was precisely doubled in dimension of 1/2 scaled model, except standpipe height.

(a) Analysis Model for Large Flow

(b) Analysis Model for Small Flow

Figure 3.2-1 Analysis Models for 1/1 Scale

(a) Analysis Model for Large Flow

(b) Analysis Model for Small Flow

Figure 3.2-2 Analysis Models for 1/2 Scale

3.2.2 Mesh Configuration

Figure3.2-3 and 3.2-4 show mesh configurations in the vortex chamber and the total number of mesh elements for both models.

For small flow injection, the circumvolution in the vortex chamber must be stronger than the flow for large flow injection. So finer mesh configuration is employed near the wall region to properly resolve the boundary layer.

As to the consideration for scaling, mesh configuration is set as follows.
Step1) The mesh configuration of the 1/2 scaled model is set as baseline.
Step2) [
]
Step3) [
]

Additionally, the meshes around the center of vortex chamber are set to be especially [
]

Figure 3.2-3 Mesh Configurations of Vortex Chamber for 1/2 and 1/1 Scaled Model for Large Flow

Figure 3.2-4 Mesh Configurations of Vortex Chamber for 1/2 and 1/1 Scaled Model for Small Flow

3.2.3 Specification of the CFD Analysis

The CFD analysis was conducted on the following specification.

- **CFD Software** : Fluent Ver.6.3.26
  (A commercial CFD package, developed by ANSYS.Inc)
- **Time Dependence**: Steady-State Calculation
- **Type of Fluid** : Incompressible Viscous Fluid
- **Viscous Model** : [ ] (See **note1**)
- **Near-Wall Treatment** : [ ] (See **note2**)
- **Cavitation Model** : [ ] (See **note3**)
- **Spacial Discretization** :
  - Momentum Equation : [ ] (See **note4**)
  - Pressure Term : [ ] (See **note5**)
  - Turbulence Source Terms : [ ] (See **note4**)
- **Other** : No Modeling for Dissolved Nitrogen Gas

(note1) [

]

(note2) [

]

(note3) [

]

(note4) [

]

(note5) [

]

3.3 Test Case for Analysis

Among seven test cases using 1/2 scale test tank, test Case3 and 6 were selected for CFD analysis conditions to cover cavitation coefficients in wide range.

Test case 3 is the case in which the test tank has the highest initial pressure among all the test cases in order to acquire the data for high pressure designing, covering the range of small cavitation coefficients. The exhaust tank pressure was 14 psig (0.098 MPa (gage)) to simulate containment inner pressure following the blowdown phase during a large LOCA.

Test case 6 has the small pressure difference between test tank and exhaust tank in order to collect the data at large cavitation coefficient, covering the range of large cavitation coefficients.

Since the injection flow rate changes significantly during large flow injection phase, three time points at the initial stage of large flow injection, at middle stage, and at the end stage are selected for the calculation points.

On the other hand, the injection flow rate is almost constant during small injection phase, therefore, two time periods at just after the flow switching, and at middle stage of injection are selected for the calculation points.

Consequently, total 20 calculation points are summed up for 1/2 scale and 1/1 scale. Test conditions and analysis time periods for test case 3 and 6 are shown in Table 3.3-1. Figure 3.3-1 shows these calculation points plotted on the 1/2 test result.

Table 3.3-1 Test Cases and Time Points for Calculation

Figure 3.3-1 Calculation Points on Flow Characteristics of Flow Damper

3.4 Boundary Conditions

Table 3.4-1 shows the boundary conditions for this CFD investigation.

Basically, the measured data obtained from 1/2 scaled test were applied for analysis models, i.e. tank pressure, tank outlet pressure, tank water level, standpipe water level. Total flow rate was calculated from the time series variation of tank water-level. Only some corrections and modifications are considered as needed for each model. For a physical reason, pressure boundaries were corrected to adjust pressure difference between 1/2 and 1/1 scaled model (See note1 and note2). In small flow injection case, for reasons of numerical calculation, flow rate were used for the inlet boundary condition instead of pressure boundary. This is because of the mass balance instability in pressure boundary condition cases (See note2). In contrast, flow rate boundary cases gave stable results. These adjustments on boundary condition contribute to obtain reasonable results for evaluation of scale effects. (See Appendix-A)

<u>For Large Flow Injection Case:</u>
- Inlet Boundary Condition: Tank Pressure (See **note1**)
- Outlet Boundary Condition: Tank Outlet Pressure (See **note2**)

<u>For Small Flow Injection Case:</u>
- Inlet Boundary Condition:
  - > Standpipe: Inlet Flow Rate (obtained by the time series variation of standpipe water-level)
  - > Small flow pipe: Inlet Flow Rate
- Outlet Boundary Condition: Tank Outlet Pressure (See **note2**)

(note1) An Inlet boundary pressure at standpipe side is corrected as follows.
The correction pressure value equivalent to a water-level difference between the height of water surface and the height of outlet pipe center is added to the measured inlet boundary pressure, i.e. tank pressure. This is because “Gravity Term” is neglected in this calculation. This correction is applied to both 1/1 and 1/2 scaled analysis model.

(note2) An outlet boundary pressure is corrected as follows.
In 1/1 scaled model, the correction pressure value equivalent to the level difference of outlet pipe center between 1/1 and 1/2 scaled model is subtracted from the measured tank outlet pressure. This is because of the level difference of outlet pipe center between 1/1(the actual ACC) and 1/2 scaled test. On the other hand, in 1/2 scaled model, the measured pressure (gauge pressure) is applied as outlet boundary condition without any corrections.

<u>**Reference Pressure and Temperature for Fluid Properties Calculation (Such as viscosity, density, and saturated pressure etc.)**</u>
- Temperature : The measured data shown in Table 3.4-1 (Constant value)
- Pressure : Sum of the following pressure values
  1) Calculated pressure (gauge pressure)
  2) Atmospheric pressure
  3) Differential pressure equivalent to level difference between the upper surface of a vortex chamber and an outlet pipe center

(In order to improve the pressure prediction accuracy on neighborhood of vortex chamber upper surface, where cavitation is likely to induce, the 3rd item of reference pressure is taken into account.)

Table 3.4-1 Boundary Condition Data for Calculation

3.5 Analysis Results

First, the applicability of CFD to the ACC is evaluated. Then CFD is used to evaluate scale effect between 1/2 and 1/1 scale model.

3.5.1 Applicability of CFD to ACC

The applicability of CFD to the ACC is assessed by comparison of the results of the 1/2 scaled test and its analysis such as flow pattern and C_v value.

(1) Flow pattern of 1/2 scale model

Flow patterns of stream lines and flow vector obtained by CFD are compared with the 1/5 scaled visualized test (Ref.1) on the vortex chamber shown in Figure 3.5-1. The chosen CFD case is comparatively similar to the test case shown in the figure.

In large flow condition, the CFD results show that the flow from the standpipe and small flow pipe joins together at the meeting point and the confluent flow flows out to the flow nozzle without a strong vortex, which is similar to the flow pattern of the test. On the other hand, in small flow condition, the CFD results show that the flow from the small flow pipe flows out to the outlet nozzle with a vortex in the vortex chamber, which is also similar to the flow pattern of the tests.

Therefore, since it is confirmed that the flow pattern in flow damper of 1/2 scaled model analysis is very similar to the one of the visualized 1/5 scaled model test, the CFD could be applied to the evaluation of scale effect.

(2) Relationship between flow rate coefficient and cavitation factor

The relationship between flow rate coefficient (C_v) and cavitation factor (σ_v) of the CFD and test result is shown in Figure 3.5-2. The characteristic equations obtained by the test results themselves include the width of the uncertainty of the tests regarding instrumental uncertainty and dispersion deviation (Ref.2). The width is shown by broken line in Figure 3.5-2.

In the large flow condition, the CFD results agree with the test results regarding tendency of flow rate coefficient with cavitation factor, that is to say, the flow rate coefficient decreases as the cavitation factor decreases.

In the small flow condition, the CFD results agree with the test results regarding the tendency of flow rate coefficient with cavitation factor which means flow rate coefficient is almost constant in the wide range of cavitation factor.

In addition, it seems that the flow rate coefficient of CFD results is almost within the width of instrumental uncertainty and dispersion deviation of test data under both flow rate conditions.

Therefore, the CFD is acceptable to evaluate the scale effect between 1/2 and 1/1 scale models of the ACC using flow pattern and C_v value on 1/2 scale model.

Figure 3.5-1(a) Flow Pattern in Vortex Chamber (Large Flow)

Figure 3.5-1(b) Flow Pattern in Vortex Chamber (Small Flow)

Figure 3.5-2 Comparison between Test Results and Calculation Results for Flow Rate Coefficient

3.5.2 Evaluation of Scale Effect Due to CFD

The scale effect is evaluated by comparison of the CFD results between 1/2 and 1/1 scaled model such as flow pattern and C_v value.

(1) Comparison of flow pattern

The compared results of CFD of the 1/2 and 1/1 scale model with those of the 1/1 scale are shown in Figure 3.5-3 to Figure 3.5-12 about flow pattern.

The CFD results of flow patterns in both large and small flow conditions for 1/1 scaled model is very similar to that of the 1/2 scaled model regarding static pressure, flow vector and void fraction as follows.

Large flow injection

The flow from the standpipe and the small flow pipe joins together at meeting point and the confluent flow flows out to the outlet nozzle without strong vortex in both scaled model.

Small flow injection

A strong vortex is occurred in the vortex chamber so that a void is generated at the center of the vortex. While void fraction generated by cavitation has a little difference between 1/2 and 1/1 scaled model, the generated void pattern such as the portion of cavitation of the 1/2 scale model is similar to that of the 1/1 scaled model.

Therefore, it is concluded that the flow pattern of CFD results are almost same to each other between 1/2 and 1/1 scaled mode.

(2) Comparison of relationship between flow rate coefficient and cavitation factor

The relationship between flow rate coefficient and cavitation factor of the CFD for the 1/2 and 1/1 scale models. The test result is shown in Figure 3.5-13.
In both large and small flow conditions, the CFD results of 1/1 scaled model agree with 1/2 scaled model regarding tendency between flow rate coefficient and cavitation factor as shown in Figure 3.5-13.

The scale effect is evaluated by Figure 3.5-14 with the abscissa of scale and the ordinate of C_v value. The figure shows that the tendency of C_v affected by scale seems different among the test case. In addition, CFD results have dispersion between 1/2 and 1/1 scaled model. Therefore this difference between 1/2 and 1/1 scaled model should be evaluated statistically to understand whether the difference is significant or not. The evaluation is shown in section 3.5.3.

Figure 3.5-3(a) Flow Pattern in Vortex Chamber (Case 3 Large Flow 5sec)

Figure 3.5-3(b) Flow Pattern in Flow Damper (Case 3 Large Flow 5sec)

Figure 3.5-4(a) Flow Pattern in Vortex Chamber (Case 3 Large Flow 20sec)

Figure 3.5-4(b) Flow Pattern in Flow Damper (Case 3 Large Flow 20sec)

Figure 3.5-5(a) Flow Pattern in Vortex Chamber (Case 3 Large Flow 34sec)

Figure 3.5-5(b) Flow Pattern in Flow Damper (Case 3 Large Flow 34sec)

Figure 3.5-6(a) Flow Pattern in Vortex Chamber (Case 6 Large Flow 5sec)

Figure 3.5-6(b) Flow Pattern in Flow Damper (Case 6 Large Flow 5sec)

Figure 3.5-7(a) Flow Pattern in Vortex Chamber (Case 6 Large Flow 20sec)

Figure 3.5-7(b) Flow Pattern in Flow Damper (Case 6 Large Flow 20sec)

Figure 3.5-8(a) Flow Pattern in Vortex Chamber (Case 6 Large Flow 50sec)

Figure 3.5-8(b) Flow Pattern in Flow Damper (Case 6 Large Flow 50sec)

Figure 3.5-9(a) Flow Pattern in Vortex Chamber (Case 3 Small Flow 43sec)

Figure 3.5-9(b) Flow Pattern in Flow Damper (Case 3 Small Flow 43sec)

Figure 3.5-10(a) Flow Pattern in Vortex Chamber (Case 3 Small Flow 100sec)

Figure 3.5-10(b) Flow Pattern in Flow Damper (Case 3 Small Flow 100sec)

Figure 3.5-11(a) Flow Pattern in Vortex Chamber (Case 6 Small Flow 82sec)

Figure 3.5-11(b) Flow Pattern in Flow Damper (Case 6 Small Flow 82sec)

Figure 3.5-12(a) Flow Pattern in Vortex Chamber (Case 6 Small Flow 200sec)

Figure 3.5-12(b) Flow Pattern in Flow Damper (Case 6 Small Flow 200sec)

Figure 3.5-13 Comparison between 1/2 and 1/1 Scale of CFD Result

Figure 3.5-14(a) Relationship between Flow Rate Coefficient and Scale (Large Flow)

Figure 3.5-14(b) Relationship between Flow Rate Coefficient and Scale (Small Flow)

3.5.3 Evaluation of significant difference between 1/2 and 1/1 scale

CFD analysis results showed a little difference for the C_v value between 1/2 and 1/1 scaled model as shown in Figures 3.5-13 and 3.5-14. It is unclear whether the difference is significant or just derived from the dispersion of the CFD results because both C_v value of 1/2 and 1/1 scale model obtained by CFD has dispersion in comparison with characteristic equation.

Therefore the significant difference test for the average C_v value of both scale model is performed to evaluate whether the difference is significant or not with [ ] of significant level which is corresponding to [ ] confidence level.

The result of significant difference test for large and small flow rate is shown in Table 3.5-1 and 3.5-2. Since C_v value has dependency to cavitation factor σ_v, non dimensional C_v is used and the difference of average value of non- dimensional C_v is evaluated by significant difference test.

Student's t distribution for [ ] of significant level is shown in Table 3.5-3.
The calculated t of both large and small flow injections are less than student's t value.

Therefore, it is concluded that the difference between the average C_v of the 1/2 scale model with that of the 1/1 scale model obtained by CFD is insignificant due to significant difference test with [ ] of significant level.

Table 3.5-1 Result of Significant Difference Test for Large Flow

Table 3.5-2 Result of Significant Difference Test for Small Flow

Table 3.5-3 Student's t Distribution for Significant Level

4.0 CONCLUSIONS

Steady-state CFD calculation with cavitation model option was conducted, using the 1/2 and 1/1 scaled models, to evaluate the validity of the current performance evaluation, i.e. extrapolating of the experiment data with the scaled model and of the characteristic equations.
The conditions of CFD calculation include both large and small flow injections with the wide range of cavitation factor.

Conclusions of the CFD results and the evaluations are shown below.

- Flow pattern in the flow damper of the 1/2 scaled model analysis is very similar to that of the visualized 1/5 scaled model test.
- The CFD prediction is feasible for large and small characteristic evaluations. In 1/2 scaled model analysis, the correlation between C_v and σ_v reasonably matched with the measured data.
- Thus, the CFD is applicable to evaluate the scale effect of the ACC.
- Flow pattern of 1/1 scaled model analysis in the flow damper and the injection piping is very similar to the one of 1/2 scaled model analysis.
- CFD analysis results showed that “scaling” has a little influence on the flow rate coefficient from comparison of the calculated data in 1/2 scaled model and 1/1 scaled one.
- The little differences of C_v value obtained by CFD analysis results between 1/2 and 1/1 scaled model is insignificant by significant difference test with [ ] of significant level.
- Thus, the validity of the current evaluation approach (Ref.1) for the ACC performance, i.e. an extrapolation from the scaled model experiment and the characteristic equations, is uncontradicted due to comparison of the calculated data in 1/2 scaled model and 1/1 scaled one.

5.0 REFERENCES

1) Topical Report “The Advanced Accumulator”, MUAP-07001 Revision 2
2) Large Break LOCA Code Applicability Report for US-APWR, MUAP-07011 Revision 0
3) ANSYS Fluent, FLUENT 6.3 Documentation, User’s Guide

Appendix-A

A Consideration for Boundary Condition in Small Flow Injection

A-1 Introduction

In the first plan, the analysis boundary conditions for large and small flow injections are assumed to be given by pressure boundary. However, for small flow injection, flow rate was used for the boundary condition instead of pressure boundary for reasons of numerical calculation.

Each small flow injection condition in this analysis is the instantaneous state at certain time in the transient flow where flow injection is gradually reduced. In small flow injection, the inflow from small flow pipe side gains greater influence to the flow in the vortex chamber due to the reduction of the inflow from standpipe side. To evaluate the instantaneous flow situations, steady-state simulation was conducted by using the measured flow data at corresponding time as boundary conditions.

As will hereinafter be described in detail, employing pressure boundary resulted in mass balance instability. Thus the solutions provided by the calculation using pressure boundary were not employed for evaluation. In contrast, the solutions provided by the calculation using flow rate boundary gave the stable solutions.

The flow rate coefficient of each solution is approximately matched with the value of the measured data. For this reason, this analytical approach is assumed to have a capability to simulate the flow in small flow injection reasonably, and provide adequate solutions to evaluate the scale effects.

The history of the above change on boundary condition is shown in this section.

A-2 History of the Boundary Condition Change

(1) Calculation with pressure boundary

Figure A-1 shows typical results with pressure boundary about “Mass flow instability at outlet” and “Mass balance”. Also Figure A-1 shows the flow situation in the vortex chamber at the pressure boundary case. These results indicate that the exit flow tends to be unstable, which results in mass balance instability.

According to the results, the following two possible reasons were presumed.

a) Fluctuation of a vortex core in the vortex chamber
(which causes mass balance instability)

b) An inflow occurrence from the standpipe inlet

(which causes an inflow instability from the small flow pipe inlet)

In situations comparing non-physical with the actual flow situation, countermeasures were conducted in order to prevent the situation, such as mesh study and shutoff of inflow from the inlet of the standpipe (i.e. to set a solid wall at the inlet of standpipe).

The calculation result with the countermeasures is shown in Figure A-2. The band of flow fluctuation was relatively improved, but still unstable. Also mass imbalance was found in some cases. This result was assumed that vortex core instability is still remaining.

(2) Change of boundary condition at inlet of standpipe to mass flow boundary

In response to the results of (1), mass flow boundary was set at the inlet of standpipe, in order to stabilize the calculation. Typical results are shown in Figure A-3.

According to the results, the band of flow fluctuation was relatively improved, but still unstable and mass imbalance was found in some cases.

(3) Change of boundary condition at inlets of standpipe and small flow pipe to mass flow boundary

In response to the results of (2), mass flow boundaries were set at the inlet of both standpipe and small flow pipe. Typical results are shown in Figure A-4.

According to the results, although an inlet pressure fluctuation occurs at the early steps, the solution settles into stable situation as iteration number increases.

A-3 Possible Reasons for Mass Imbalance in Pressure Boundary Condition

One possible reason may be pressure fluctuation due to the vortex core movement.

[

]

Figure A-1 Typical Results with Pressure Boundary

Figure A-2 Calculation Result with Wall Condition at Inlet of Standpipe as Boundary Condition

Figure A-3 Calculation Result with Flow Condition at Inlet of Standpipe as Boundary Condition

Figure A-4 Calculation Result with Flow Condition at Both Standpipe and Small Flow Pipe Inlet as Boundary Condition