


MITSUBISHI HEAVY INDUSTRIES, LTD.
16-5, KONAN 2-CHOME, MINATO-KU
TOKYO, JAPAN

June 25, 2010

Document Control Desk
U.S. Nuclear Regulatory Commission
Washington, DC 20555-0001

Attention: Mr. Jeffrey A. Ciocco

Docket No. 52-021
MHI Ref: UAP-HF-10180

Subject: MHI's Responses to NRC's Request for Additional Information Technical Report CFD Analysis for Advanced Accumulator MUAP-09025

Reference: [1] "Request for Additional Information Technical Report CFD Analysis for Advanced Accumulator MUAP-09025" dated May 14, 2010.

With this letter, Mitsubishi Heavy Industries, LTD. ("MHI") transmits to the U.S. Nuclear Regulatory Commission ("NRC") the document entitled "MHI's Responses to NRC's Request for Additional Information Technical Report CFD Analysis for Advanced Accumulator MUAP-09025" dated May 14, 2010.

Enclosed are the responses to No.54, 55 except 55 f) and 57 that are contained within Enclosure 2 and 3.

As indicated in the enclosed materials, this document contains information that MHI considers proprietary, and therefore should be withheld from public disclosure pursuant to 10 C.F.R. § 2.390 (a)(4) as trade secrets and commercial or financial information which is privileged or confidential. A non-proprietary version of the document is also being submitted with the information identified as proprietary redacted and replaced by the designation "[]".

This letter includes a copy of the proprietary version (Enclosure 2), a copy of the non-proprietary version (Enclosure 3), and the Affidavit of Yoshiki Ogata (Enclosure 1) which identifies the reasons MHI respectfully requests that all materials designated as "Proprietary" in Enclosure 2 be withheld from public disclosure pursuant to 10 C.F.R. § 2.390 (a)(4).

Please contact Dr. C. Keith Paulson, Senior Technical Manager, Mitsubishi Nuclear Energy Systems, Inc. if the NRC has questions concerning any aspect of the submittal. His contact information is below.

Sincerely,



Yoshiki Ogata,
General Manager- APWR Promoting Department
Mitsubishi Heavy Industries, LTD.

DOB1
NRC

Enclosures:

- 1 - Affidavit of Yoshiki Ogata
- 2 - MHI's Responses to NRC's Request for Additional Information Technical Report CFD Analysis for Advanced Accumulator MUAP-09025 (proprietary)
- 3 - MHI's Responses to NRC's Request for Additional Information Technical Report CFD Analysis for Advanced Accumulator MUAP-09025 (non-proprietary)

CC: J. A. Ciocco
C. K. Paulson

Contact Information

C. Keith Paulson, Senior Technical Manager
Mitsubishi Nuclear Energy Systems, Inc.
300 Oxford Drive, Suite 301
Monroeville, PA 15146
E-mail: ckpaulson@mnes-us.com
Telephone: (412) 373-6466

ENCLOSURE 1

Docket No. 52-021
MHI Ref: UAP-HF-10180

MITSUBISHI HEAVY INDUSTRIES, LTD.

AFFIDAVIT

I, Yoshiki Ogata, state as follows:

1. I am General Manager, APWR Promoting Department, of Mitsubishi Heavy Industries, LTD ("MHI"), and have been delegated the function of reviewing MHI's US-APWR documentation to determine whether it contains information that should be withheld from public disclosure pursuant to 10 C.F.R. § 2.390 (a)(4) as trade secrets and commercial or financial information which is privileged or confidential.
2. In accordance with my responsibilities, I have reviewed the enclosed document entitled "MHI's Responses to NRC's Request for Additional Information Technical Report CFD Analysis for Advanced Accumulator MUAP-09025" dated June 2010, and have determined that portions of the document contain proprietary information that should be withheld from public disclosure. Those pages containing proprietary information are identified with the label "Proprietary" on the top of the page and the proprietary information has been bracketed with an open and closed bracket as shown here "[]". The first page of the document indicates that all information identified as "Proprietary" should be withheld from public disclosure pursuant to 10 C.F.R. § 2.390 (a)(4).
3. The information identified as proprietary in the enclosed document has in the past been, and will continue to be, held in confidence by MHI and its disclosure outside the company is limited to regulatory bodies, customers and potential customers, and their agents, suppliers, and licensees, and others with a legitimate need for the information, and is always subject to suitable measures to protect it from unauthorized use or disclosure.
4. The basis for holding the referenced information confidential is that it describes the unique design of the Advanced Accumulator developed by MHI and not used in the exact form by any of MHI's competitors. This information was developed at significant cost to MHI, since it required the performance of Research and Development and detailed design for its software and hardware extending over several years.
5. The referenced information is being furnished to the Nuclear Regulatory Commission ("NRC") in confidence and solely for the purpose of information to the NRC staff.
6. The referenced information is not available in public sources and could not be gathered readily from other publicly available information. Other than through the provisions in paragraph 3 above, MHI knows of no way the information could be lawfully acquired by organizations or individuals outside of MHI.
7. Public disclosure of the referenced information would assist competitors of MH in their design of new nuclear power plants without incurring the costs or risks associated with the design and testing of the subject systems. Therefore, disclosure of the information contained in the referenced document would have the following negative impacts on the competitive position of MH in the U.S. nuclear plant market:

- A. Loss of competitive advantage due to the costs associated with development and testing of the Advanced Accumulator. Providing public access to such information permits competitors to duplicate or mimic the Advanced Accumulator design without incurring the associated costs.
- B. Loss of competitive advantage of the US-APWR created by benefits of enhanced plant safety, and reduced operation and maintenance costs associated with the Advanced Accumulator.

I declare under penalty of perjury that the foregoing affidavit and the matters stated therein are true and correct to the best of my knowledge, information and belief.

Executed on this 25th day of June, 2010.



Yoshiki Ogata,
General Manager- APWR Promoting Department
Mitsubishi Heavy Industries, LTD.

Enclosure 3

UAP-HF-10180
Docket No. 52-021

**MHI's Responses to NRC's Request
for Additional Information**

**Technical Report CFD Analysis for Advanced Accumulator
MUAP-09025**

June 2010
(Non-Proprietary)

RAI for CFD Analyses for Advanced Accumulator, MUAP-09025-P (R0)

In its letter of December 11, 2009, MHI submitted the computational fluid dynamics (CFD) calculations for ½ - and 1/1- scale advanced accumulator (ACC) facilities for the large-flow and small-flow phases, respectively. The objective of these analyses is to assess the applicability of the advance accumulator characteristics equation developed from ½ scale facility to full scale facility. This will require quantification of scale effect in the characteristic equations, including the effect of the vortex size (vortex chamber size) on the flow resistance and cavitation. As a results of its review of the CFD analysis report, the NRC staff requests further clarification on the following items.

RAI 54.

In the CFD model development with the FLUENT code, a few assumptions were made that need to be explained. Some of these are related to pressure boundary conditions.

- a) In Section 3.4 of the report, Note 1 indicated that the inlet boundary pressure at the standpipe side is corrected by the hydrostatic pressure related to the difference in the height of water surface and the height of outlet pipe center for full scale and half scale facilities for large flow phase because the "Gravity Term" is neglected in the calculation. The staff believes that even if the pressure drop is much higher than the gravity term, it should be left in the analyses.

Why is this gravity term neglected?

- b) As explained in the FLUENT manual, the pressure field P_s' and the user pressure inputs include the hydrostatic head, $\rho_0 gx$. That is, the pressure in FLUENT is defined as:

$$P_s' = P_s - \rho_0 gx$$

or

$$\frac{\partial P_s'}{\partial x} = \frac{\partial P_s}{\partial x} - \rho_0 g$$

where ρ_0 is the operating density as specified through the input.

This definition allows part of the hydrostatic head to be included in the body force term, $(\rho - \rho_0)g$, and excluded from the pressure calculation when the density is uniform and constant. Therefore, the inputs of pressure should not include hydrostatic pressure differences and reports of pressure P_s' will not show any influence of the hydrostatic pressure.

Please explain how your approach of specifying pressures at the inlet and the outlet is consistent with the FLUENT formulation as explained here.

- c) In the boundary condition section (Sect. 3.4, Note 2) it is mentioned that the outlet

boundary conditions for the full scale is specified by subtracting the hydraulic pressure equivalent of height difference between the outlets for full scale and half scale facilities. For the 1/2 scale facility, the measured pressure is applied as outlet boundary condition without any correction.

- Please explain why this correction is needed for full scale facility calculations?
 - What are the differences in vertical dimensions of full scale and half scale facilities?
- d) Please show the difference in pressure values for the inlet and exit boundaries after the corrections were made for different cases.

Response

- a) The sentence of Note 1 in Section 3.4 of the report is emended to correct depiction by revising the report as following, from "Gravity Term is neglected in the calculation" to "Gravity Term is included into Pressure Term in the calculation".

In order to shorten the calculation time and reduce the round-off errors, the gravity term has been included into the pressure term in the momentum equation in this CFD study. Since there is a different between the height of the inlet boundary and the outlet boundary, proper correction for measured pressure of inlet tank has to be performed. The correction method has been explained in RAI54-c).

- b) In the current CFD study, the gravity term has been included into the pressure term in the momentum equation as that has been indicated in the FLUENT manual. Based on this method, the proper way to specify the inlet and the outlet boundary pressure conditions has been explained in RAI54-c).

c)

1. Correction method for boundary pressure values

In fact the outlet boundary condition in 1/1 scale model has been specified by using the measured pressure values in 1/2 scale test facility. Thus, Note 1 and Note 2 in Section 3.4 should be modified as follows.

(note1) An Inlet boundary pressure at tank 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 included into Pressure Term in the calculation.

Here the inlet boundary pressure in the 1/1 scale model has been specified by utilizing the same pressure value in the 1/2 scale model. However, due to the fact that outlet pipe elevation in the 1/1 scale model is higher than that in the 1/2 scale model, hydraulic head caused by the elevation difference between the tank water surface and the center of outlet pipe in 1/2 scale model is different than that in the 1/1 scale model. Thus, proper corrections should be made in order to match the inlet/outlet differential pressure in the 1/2 scale model.

*(note2) An outlet boundary pressure is corrected as follows.
The reference pressure location has been specified at the outlet pipe. Thus, measured pressure data have been utilized as the outlet boundary pressure without correction.
Same corrected pressure measurement data have been applied in the 1/1 scale model.*

Above modifications have been reflected in the revised report.

Details of the correction method are explained as follows. Since the gravity term has been included into the pressure term in these analyses, corrections on boundary pressure values in the 1/2 scale model should be performed.

Measured inlet pressure does not contain the water hydrostatic head, thus

$$P'_1 = P_1$$

On the other hand, measured outlet pressure contains the hydrostatic head caused by the water height difference between the water surface in tank and the center of outlet pipe. Thus,

$$P'_2 = P_2 - \rho_0 g z_{1/2}$$

Where,

P'_1 : Tank pressure excluding the hydrostatic head

P_1 : Measured tank pressure

P'_2 : Outlet pipe pressure excluding the hydrostatic head

P_2 : Measured outlet pipe pressure

ρ_0 : Operating density

g : Gravitational acceleration

$z_{1/2}$: Water-level difference between the height of water surface and the height of outlet pipe center of 1/2 scale model.

The reference pressure location in these analyses has been specified at the outlet pipe. Thus, input values of boundary conditions should be corrected by considering the hydrostatic head caused by water height difference between the tank water surface and the outlet pipe. In this process, the inlet/outlet differential pressure should match the difference between P'_1 and P'_2 in above equations.

$$P''_1 = P'_1 + \rho_0 g z_{1/2} = P_1 + \rho_0 g z_{1/2}$$

$$P''_2 = P'_2 + \rho_0 g z_{1/2} = P_2$$

Where,

P''_1 : Input pressure for inlet tank

P''_2 : Outlet pressure for outlet pipe

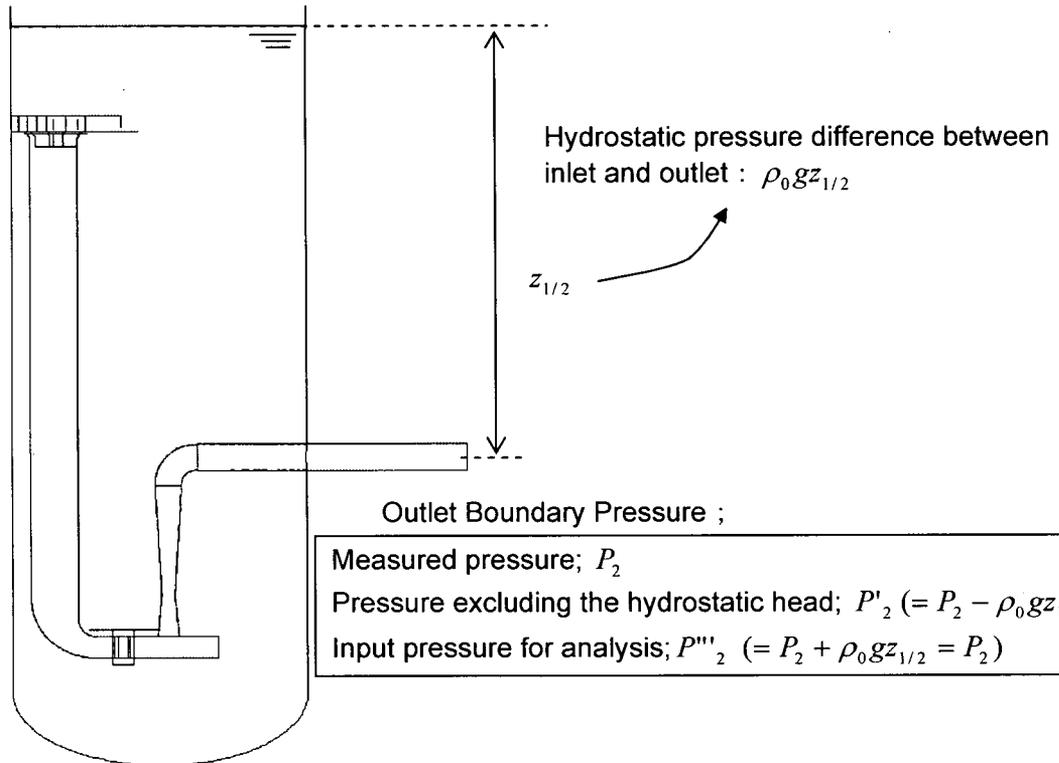
The outlet pipe elevation in the 1/1 scale model is higher than that in the 1/2 scale model. Hydraulic head caused by the elevation difference between the tank water surface and the center of outlet pipe in 1/2 scale model is different than that in the 1/1 scale model. Thus, same boundary pressure values as that in the 1/2 scale model have been specified in the 1/1 scale model in order to match the inlet/outlet differential pressure (which is the driving force of the flow) in the 1/2 scale model. Figure 1-1 explains this in detail.

2. The height difference between 1/1 scale model and 1/2 scale model

The height of standpipe in 1/2 scale model is same as that in the 1/1 scale model. However, the elevation of outlet pipe in the 1/1 scale model is twice of that in the 1/2 scale model. From the top of vortex chamber, the elevation of outlet pipe in the 1/1 scale model is [] m, while the elevation of outlet pipe in the 1/2 scale model is [] m.

Inlet Boundary Pressure:

Measured pressure; P_1
 Pressure excluding the hydrostatic head; $P'_1 (= P_1)$
 Input pressure for analysis; $P''_1 (= P'_1 + \rho_0 g z_{1/2})$



Note: Since "Gravity Term" is included into Pressure Term in the calculation, same inlet/outlet boundary pressure values as that in the 1/2 scale model have been specified in the 1/1 scale model

Figure 1-1 Boundary Pressure Setup Method in 1/2 scale Model

- d) Pressure inlet boundary condition has been utilized in large injection flow CFD analysis models. Corrected inlet boundary pressure values for all simulation cases are summarized in Table 1.1. This table also shows the measured pressures as reference values. As that has been mentioned in RAI54-c), identical differential pressures of the inlet boundary and the outlet boundary have been specified for the 1/1 scale model and for the 1/2 scale model in order to achieve the same driving force for the flow.

Table 1.1 Boundary Conditions for Large Injection Flow CFD Models

RAI 55.

The following items need further clarification regarding the MHI CFD results.

- a) The results provided in the log-log coordinate plots for the flow rate coefficient and the cavitation factor do not provide actual difference due to scale. The flow rate coefficient in the log form masks the real differences.

Please provide a table with the predicted flow rates (for the large-flow phase) and tank pressures for the small-flow rate phase. Also, provide these types of tables for assessment of the CFD prediction with the test data from the ½ scale facility.

- b) The inlet mass flow rates were used as boundary conditions for the small flow rate cases. What are the corresponding pressure differences between the inlet and the outlet pressures, and how do they compare with the measured values for the ½ scale data?
- c) If the full scale facility has no scale distortion compared to the ½ scale facility, the full scale facility should have 4 times the flow rate of the ½ scale. In the CFD analyses, the boundary conditions for the small-flow cases are the flow rate, and as indicated in the footnote to Table 3.4-1, the flow rate used for the full scale model is four times the flow of the ½ scale facility.

With the quadrupled flow rates specified in the full scale models, how do the CFD analyses for the small flow phase address the scaling effect? How can these analyses be used to quantify scale distortions in the characteristic equations?

- d) A derivation of relationship between flow rate coefficient and cavitation parameter indicate that there are only two free variables, i.e., the discharge velocity and discharge pressure as shown here in log-log form:

$$\log(\sigma_v) = 2 \log(c_v) + \log\left(\frac{P_D - P_v}{\rho V_D^2}\right)$$

If V_D and P_D are the same for the ½-scale facility and 1/1 full scale, the characteristic relationship will be the same for two facilities. These are specified as boundary conditions in the CFD analyses.

How can the effect of scaling on characteristic equation be assessed based on these analyses?

- e) Figures 3.5-3(a) to 3.5-5(a) show the velocities in the vortex chamber for the large-flow cases. Why is the velocity higher for the ½-scale facility than in the 1/1 scale model?
- f) In the 2 cases (Test cases 3 and 6 in Table 3.4-1) MHI chose to analyze, please provide figures that can show when a cavitation starts and stops. In addition, please provide the actual values of void fraction in the vortex chamber for the small-flow regime.
- g) Please provide a section that shows a quantitative assessment of scale effect on the characteristic equations based on the CFD analyses.

Response

- a) Predicted and measured cavitation factors and flow rate coefficients have been summarized in Table 2-1 and Table 2-2. These tables also show the inlet pressure, outlet pressure, and outlet injection flow rate that have been applied as the boundary conditions for CFD simulations.

In large injection flow cases, measured pressure values have been applied as the inlet and outlet boundary conditions. As that has been described in RAI54-c, these pressure values have been adjusted by choosing the elevation of outlet pipe as the reference level. On the other hand, in small injection flow cases, flow rate has been applied as the inlet boundary condition and pressure has been applied as the outlet boundary condition. Values that have been given as the simulation boundary conditions and values that have been calculated by CFD simulation are marked with different colors in Table 2-1 and Table 2-2.

Table2-1 Comparison of Parameters in 1/2 Scale Model



Table2-2 Comparison of Parameters in CFD Simulations



- b) Table 2-1 and Table 2-2 in RAI55-a show the pressure difference between inlet and outlet.

Flow rate in the standpipe and in the small flow pipe have not been obtained in large flow injection tests, while tank pressure has been measured in these tests. Thus only measured tank pressure can be specified as the boundary condition in large flow injection CFD analysis models.

On the other hand, not only the tank pressure but also the flow rate in the standpipe and the small flow pipe have been obtained in small flow injection tests. Thus, either measured tank pressure or measured flow rate can be chosen as the inlet boundary condition in small flow injection CFD analysis models.

The cavitation factor σ_v and the flow rate coefficient C_v depend on the flow velocity, as well as on the pressure difference between inlet and outlet. In CFD analysis models, either flow rate or pressure can be specified as the boundary condition. Values of the remaining parameter will be determined by the CFD calculation. σ_v and C_v then can be calculated from the CFD analysis results. Objective of these analyses are to evaluate the scaling effect between 1/2 scale model and 1/1 scale model. This scaling effect can be properly evaluated if same method has been followed for calculating σ_v and C_v in 1/2 scale model and in 1/1 scale model. On the other words, scaling effect can be illustrated through the σ_v and C_v calculated from the pressure simulation results with flow rate specified as the boundary condition, or through the σ_v and C_v calculated from the flow rate simulation results with pressure specified as the boundary condition. Through significant difference tests it has been confirmed that there is no significant difference between these calculated values in 1/2 scale model and in 1/1 scale model. Thus, it is believed that scaling effect does not exist.

Based on the following reason, flow rate has been specified as the inlet boundary condition in small flow injection CFD analysis models. In cases that pressure has been specified as inlet boundary condition, CFD simulation will spend large CPU time in order to reach converged solutions. On the other hand, reasonable calculation time is needed in cases that flow rate has been specified as the inlet boundary condition. Thus, flow rate has been chosen as the inlet boundary conditions in order to finish all necessary calculations on time.

In small flow injection cases, pressure values have been calculated by CFD simulation. The pressure difference between inlet and outlet, CFD calculated pressure values for 1/2 scale model and for 1/1 model are different from each other. However, through significant difference test it has been confirmed that there is no significant difference in calculated values.

- c) As that has been explained in RAI55-b), flow rate has been specified as the inlet boundary condition in small flow injection CFD analysis models.

Inlet boundary condition of 1/2 scale model has been specified by using the inlet flow rate calculated from measured tank and standpipe water levels. Since not only the tank pressure but also the flow rate in the standpipe and the small flow pipe have been obtained in small flow injection tests, it is possible to specify flow rate of the standpipe and the small pipe as the inlet boundary condition.

Inlet flow rate in the 1/1 scale model are not known due to such data are not obtained in tests. Considering that the pipe cross-section area in the 1/1 scale model is 4 times of that in the 1/2 scale model, we multiply the inlet flow rates in the 1/2 scale model by 4 to obtain the inlet flow rates in the 1/1 scale model. Here, averaged inlet flow velocity in the 1/2 scale model has been assumed to be same as that in the 1/1 scale model.

These analyses intend to evaluate the cavitation factor and the flow coefficient C_v , which are functions of the outlet pipe flow velocity and the pressure difference between the inlet tank and the outlet pipe. C_v can be expressed as:

$$\sigma_v = \frac{P_D + P_{at} - P_v}{P_{gas} - \left(P_D + \frac{1}{2} \rho v_D^2 + \rho g H \right)}$$

$$C_v = \sqrt{\frac{V_D^2 \rho}{2 \left(P_g - P_D - \frac{1}{2} \rho V_D^2 + \rho g H \right)}}$$

Where,

- σ_v : Cavitation factor [-]
- C_v : Flow coefficient [-]
- V_D : Flow velocity in the outlet pipe [m/s]
- P_g : Inlet tank pressure [Pa]
- P_D : Outlet pipe pressure [Pa]
- ρ : Water density [kg/m³]
- g : Gravity [m/s²]
- H : Water-level difference between the height of inlet tank water surface and the height of outlet pipe center [m]

Among these parameters, the inlet flow rate and outlet pressure have been specified as the boundary conditions in small flow injection CFD analysis models, while the inlet tank pressure P_g and outlet pipe velocity V_D can be computed by the CFD. In the case that scaling effect does exist, CFD calculated P_g and V_D values in the 1/2 scale model will be

different from P_g and V_D values in the 1/1 scale model, even though the inlet flow rate in the 1/1 scale case has been specified as 4 times of that in the 1/2 scale model. As a result, calculated σ_v and C_v values for the 1/2 scale model and for the 1/1 scale model will be different. Through the significant difference test we found that although calculated values for the 1/2 scale model and for the 1/1 scale model do have differences, such differences are not caused by the scaling effect. Therefore, it is believed that specifying the inlet flow rate in 1/1 scale model equals to 4 times of that in 1/2 scale model will not generate any problems in evaluating the scaling effect on σ_v and C_v .

d) The cavitation factor and flow rate coefficient are related through the following equation

$$\log \sigma_v = 2 \log C_v + \log \frac{P_D - P_v}{\rho V_D^2}$$

Derivation of above equation is shown in this section.
 Definitions of all parameters are listed here.

Flow rate coefficient C_v

$$C_v = \frac{1}{\sqrt{K_D}}$$

$$K_D = \frac{(P_A + \rho g H) - (P_D + \rho V_D^2 / 2 + \rho g H')}{\rho V_D^2 / 2}$$

Cavitation factor σ_v

$$\sigma_v = \frac{P_D + P_{at} - P_v}{(P_A + \rho g H) - (P_D + \rho V_D^2 / 2 + \rho g H')}$$



Explain of pressure of each part and elevation difference

Here,

- K_D : Form loss coefficient of in flow damper
- P_A : Tank gas space pressure [Pa (gauge)]
- ρ : Water density (kg/m³)
- g : Gravity (m/sec²)
- H : Elevation difference between liquid surface and the top of flow damper (m)
- H' : Elevation difference between the top of flow damper and measurement point of P_D (m)
- P_{at} : Atmosphere pressure (Pa)
- P_v : Vapor saturation pressure (Pa)
- P_D : Outlet pressure (Pa (gauge))*
- V_D : Outlet velocity (m/sec)*

* These parameters should be converted according to prototype scales.

From this

$$C_v = \frac{1}{\sqrt{\sigma_v \times \frac{P_D - P_{at} - P_v}{\rho V_D^2 / 2}}}$$

$$\leftrightarrow \sigma_v = C_v^2 \times \frac{P_D - P_{at} - P_v}{\rho V_D^2 / 2}$$

By taking the logarithm,

$$\leftrightarrow \log \sigma_v = \log \left(C_v^2 \times \frac{P_D - P_{at} - P_v}{\rho V_D^2 / 2} \right)$$

$$\leftrightarrow \log \sigma_v = \log C_v^2 + \log \frac{P_D - P_{at} - P_v}{\rho V_D^2 / 2}$$

$$\leftrightarrow \log \sigma_v = 2 \log C_v + \log \frac{P_D - P_{at} - P_v}{\rho V_D^2 / 2}$$

Here, constants (Pat and 2) in the second term of right hand side can be ignored since they will not affect the characteristic equation significantly

$$\log \sigma_v \cong 2 \log C_v + \log \frac{P_D - P_v}{\rho V_D^2}$$

In the above equation, no parameters are related to the facility scales. So, there will be no scaling effect if the value between outlet velocity and inlet/outlet pressure difference are same)

On the other hand, there are no discrepancies between the CFD simulation results of outlet velocity and inlet/outlet pressure difference in 1/2 scale model and in 1/1 scale model, as that have been explained in RAI55-b) and RAI55-c). Thus no scaling effect has been found in this study. In large injection flow CFD analysis models, inlet and outlet pressure have been specified as the boundary conditions, and the outlet velocity has been calculated by the CFD. In small injection flow CFD analysis models, inlet flow rate has been specified as the boundary condition, and the inlet/outlet pressure difference has been calculated by the CFD. If there is no difference between the CFD calculated parameters for the 1/2 scale model and for the 1/1 scale model, the above characteristic equation between the flow rate coefficient and the cavitation factor will be exactly same. Then it is believed that no scaling effect exists in the characteristic equation.

As that has been explained in RAI55-a, CFD simulation results show that there are some differences between the predicted outlet velocity and inlet pressure in the 1/2 scale model with those in the 1/1 scale model. However, it has been confirmed that these differences are not significant by performing the significant difference test.

From the above explanation on both the characteristic equation and the CFD simulation results, it can be concluded that there is no scaling effect in the characteristic equation.

- e) Figure 2-1 to 2-3 plot the vertical distribution of velocity magnitude in the high speed domain in the vortex chamber. These figures show there are slight differences between the velocity distributions in the vortex chamber of 1/2 scale models with that in the vortex chamber of 1/1 scale models. Figures 3.5-3(a) to 3.5-5(a) plot the vertical velocity magnitudes at the center of vortex chamber. These figures show that at some levels, flow velocity in 1/1 scale models is higher than that in the 1/2 scale models. The reason has been reported in RAI57, such that, this is caused by flow field fluctuations during the convergence process for steady-state calculations.

At some locations, flow velocity in the 1/2 scale models is higher than that in the 1/1 scale models. Same as that has been explained in the above, this was caused by the calculation convergence process and by the locations that has been chosen to perform the evaluation. This difference is not caused by the scaling effect. As has been shown in the table of RAI55-a), the overall difference between averaged flow rate is small. Therefore, it cannot be said that the velocity is higher for the 1/2 scale facility than in the 1/1 scale model.



Figure 1-1 Flow Pattern in Vortex Chamber (Case 3 Large Flow 5 sec)

Figure 1-2 Flow Pattern in Vortex Chamber (Case 3 Large Flow 20 sec)

Figure 1-3 Flow Pattern in Vortex Chamber (Case 3 Large Flow 34 sec)

- f) This response will be submitted later.

- g) The technical report will be revised to add a section that shows a quantitative assessment of scale effect on the characteristic equations based on the CFD analyses.

RAI 56.

The following documents provide best practice guidelines for the CFD analyses of nuclear reactors:

- Assessment Of Computational Fluid Dynamics (CFD) For Nuclear Reactor Safety Problems, January 2008, JT03239346, NEA/CSNI/R (2007)13
- Best Practice Guidelines for the use of CFD in Nuclear Reactor Safety Applications, NEA/CSNI/R (2007)5, 15-May-2007
- Policy of Journal of Fluids Engineering of ASME about CFD analyses. (Journaltool.asme.org/Content/JFENumAccuracy.pdf)

- a) Were these best practices used in the MHI's CFD analyses?
- b) Per the best practice guideline, the results of CFD analysis should include an estimate of numerical uncertainty and grid convergence.

Please provide an estimate of numerical uncertainty in the MHI's CFD analyses results. Please also provide results of grid convergence analyses.

- c) Have any sensitivity been performed concerning the following?
1. Turbulence Modeling
 2. Boundary conditions
 3. Grid independence and grid convergence results.
 4. Was grid convergence Index (GCI) used to assess the uncertainty?
 5. Sensitivity on order of magnitude (second order was used only in the momentum equation).
 6. Sensitivity on the type of wall function (using y^+)

Response

This response will be submitted later.

RAI 57.

Appendix A describes the reason for selecting flow boundary conditions for small flow rate phase. It is stated that there were large fluctuations in the outlet flow when steady pressure boundary conditions were applied in small flow phase.

Did the tests show outlet flow fluctuations for small flow phase? What is the reason of these fluctuations in the tests if observed, and in the CFD analyses?

Response

Revealed outlet flow rate fluctuations in current CFD steady-state analyses are caused by the solution variations per iteration during the convergence process. There is no transient effect in these fluctuations. In Appendix-A of technical report, we described that the outlet flow had fluctuations if pressure inlet condition was specified. However, the fluctuations in CFD results were again caused by the solution changes per iteration during the convergence process. They did not represent the transient fluctuations caused by time changing in the real phenomenon. The results of CFD analysis are obtained through a steady-state solver, thus, no time effects have been considered in these analyses.

It is believed that large solution fluctuations happen during the solution convergence process due to that: Strong vortex flow forms in the vortex chamber under small flow rate injection conditions. It is difficult for CFD simulation to reach a stable vortex shape. Thus, the flow resistance caused by the vortex is fluctuated in the CFD simulation results.