REQUEST FOR ADDITIONAL INFORMATION TECHNICAL REPORT
CFD ANALYSIS FOR ADVANCED ACCUMULATOR
MUAP-09025

05/13/10

US-APWR TOPICAL REPORT
THE ADVANCED ACCUMULATOR MUAP-07001

Mitsubishi Heavy Industries

Docket No. 52-021

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.

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, $\rho_o gx$. That is, the pressure in FLUENT is defined as:

      $$P_s' = P_s - \rho_o g x$$

      or

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

      where ρ_o 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, $(\rho-\rho_o)$ 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 ½ 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.

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:

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

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.

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. (J***ournal****tool.****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+)

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?