Proceedings of the 16th International Conference on Nuclear Engineering
ICONE16
May 11-15, 2008, Orlando, Florida, USA

ICONE16-48206

VALIDATION OF FLUID-STRUCTURE INTERACTION CALCULATIONS IN A LARGE-BREAK LOSS OF COOLANT ACCIDENT

Antti Timperi
VTT Technical Research Centre of Finland
POB 1000, FI-02044 VTT, Finland

Timo Pättikangas
VTT Technical Research Centre of Finland
POB 1000, FI-02044 VTT, Finland

Ismo Karppinen
VTT Technical Research Centre of Finland
POB 1000, FI-02044 VTT, Finland

Ville Lestinen
Fortum Nuclear Services Ltd
POB 100, FI-00048 FORTUM, Finland

Jukka Kähkönen
Fortum Nuclear Services Ltd
POB 100, FI-00048 FORTUM, Finland

Timo Toppila
Fortum Nuclear Services Ltd
POB 100, FI-00048 FORTUM, Finland

ABSTRACT

In the hypothetical Large-Break Loss Of Coolant Accident (LBLOCA), rapid depressurization of the reactor primary circuit causes loads on the reactor internals. This paper presents numerical simulations of a HDR experiment, where LBLOCA of a pressurized water reactor due to a sudden pipe break in the primary loop was studied. In the experiment, Fluid-Structure Interaction (FSI) phenomena caused by the flexibility of the core barrel were studied in particular.

Star-CD Computational Fluid Dynamics (CFD) code and ABAQUS structural analysis code were used for three-dimensional calculations. The MpCCI code was used for two-way coupling of the CFD and structural analysis codes in order to take FSI into account. Two-way FSI calculation was also performed with ABAQUS only by modeling water as an acoustic medium. Pressure boundary condition at the pipe break was evaluated with the system code APROS as a two-phase calculation.

Comparisons with the experiment were made for fluid pressures and break mass flow as well as for structural displacements and strains. Fairly good agreement was found between the experiment and simulation when coupling of the CFD and structural analysis codes was used. For the acoustic calculation, the results showed good agreement in the early phase of the simulation. In the late phase, structural loads were over-predicted by the acoustic calculation due to the effect of bulk flow of water which is not included in the acoustic model.

NOMENCLATURE

c	Speed of sound [m/s]
E	Elastic modulus [Pa]
p	Pressure [Pa]
T	Temperature [ºC]
t	Time [s]
β	Stiffness proportional damping
μ	Molecular viscosity [Pa·s]
ν	Poisson's ratio
ρ	Density [kg/m^3]

1. INTRODUCTION

In the hypothetical Large-Break Loss Of Coolant Accident (LBLOCA), the rapid "guillotine" break of one of the main coolant pipes of the primary circuit causes a rapid pressure drop at the break location. The pressure change propagates as a pressure wave inside the reactor pressure vessel and induces transient loads on the reactor internals.

The sudden pressure drop induces at first outward motion of the core barrel near the break, which affects the pressure in the downcomer and in the reactor core. In the downcomer, the outward motion of the core barrel partly suppresses the pressure drop. In the reactor core, the outward motion of the core barrel creates a rarefaction wave. These kinds of Fluid-

Structure Interaction (FSI) effects decrease the load on the core barrel caused by the first pressure drop after the pipe break. As a consequence of FSI, frequency of structural motion is also significantly reduced. In earlier studies, it has been found to be important to account for the two-way FSI coupling in simulations of LBLOCA [1,2]. Simulation of the event also requires evaluation of the pressure drop at the pipe break point with a method that sufficiently accounts for the two-phase phenomena near the break.

In this work, FSI calculations of LBLOCA are validated against the HDR (Heißdampfreaktor) experiments, where LBLOCA was studied in a full-scale geometry by using realistic initial conditions. Pressure boundary condition at the pipe break is calculated with the APROS system code. FSI is simulated by two-way coupling of Computational Fluid Dynamics (CFD) and structural analysis codes by using the MpCCI coupling software. Star-CD is used for fluid dynamic calculations and ABAQUS for structural analysis. In addition, FSI calculation of the HDR experiment is carried out with ABAQUS by using acoustic modeling for water.

2. HDR BLOWDOWN EXPERIMENTS

The HDR blowdown experiments were carried out in the early 1980's in Germany [2,3,4]. The test reactor was originally a working prototype of superheated steam reactor, but was after brief operation shut down and used for nuclear safety research. The test facility has therefore realistic size and construction. FSI phenomena caused by flexibility of the core barrel during the initial depressurization phase were studied in particular. One of the main emphasis in the experiments was validation of three-dimensional FSI codes.

Lay-out and main dimensions of the test facility are shown in Fig. 1. Main parameters of the experiments are compared to those of the VVER-440 type pressurized water reactor (PWR) in Table 1. As can be seen, dimensions and initial conditions of the test facility are close to those of a real PWR. However, a short break opening time, about 1 - 2 ms, was used in the experiments, whereas opening times of 10 - 15 ms or even longer have been proposed for a realistic break [5,6].

The pipe break was simulated with a double break disk system shown in Fig. 2. For measuring the mass flow rate, two drag bodies having rectangular sharp-edged drag plates were installed in the measurement ring located approximately in the middle of the nozzle. The total blockage area of the drag bodies was approximately 18.8 × 10^{-4} m^2, which resulted in a blockage of about 6 % of the free-flow cross-sectional area.

Realistic initial temperature distribution was achieved with hot and cold water loops, which were valved off just prior to the blowdown. Most of the original nozzles of the reactor were closed in order to provide clear boundary conditions for the fluid dynamic calculations. However, some of the nozzles were left open during the experiments but effect of these on the blowdown process was estimated generally small [3].

The internals of the reactor were removed and their effect was simulated with a mass ring attached to the lower end of the core barrel. The lower end of the core barrel was free and the upper end was rigidly clamped.

Blowdown experiment V32, which was the base case in the experiment series, was chosen for this work. In this experiment, the downcomer and break pipe temperature was 240 ºC and the core temperature varied axially from 308 ºC at the upper core to 283 ºC at the lower core barrel end. Subcooling in the downcomer and break pipe area was quite large in experiment V32, i.e. 78 ºC, which increased loads on the core barrel.

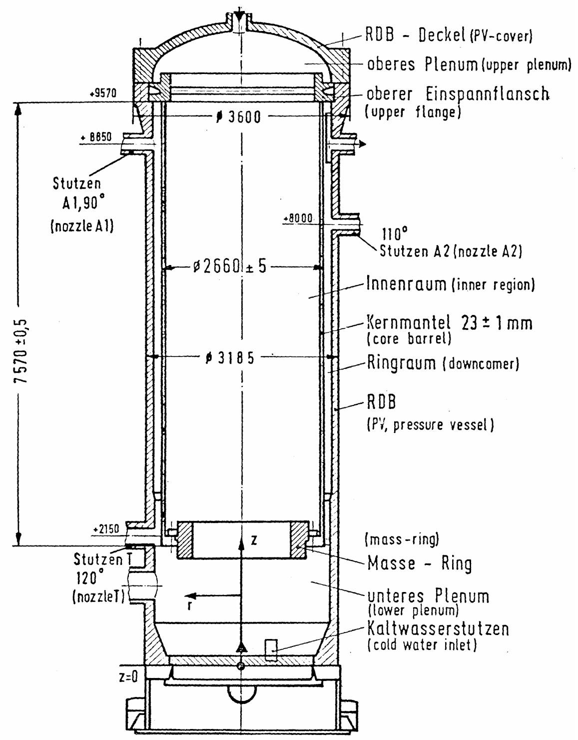

Figure 1. HDR reactor. Break occurs in nozzle A1 shown on the left. [4]

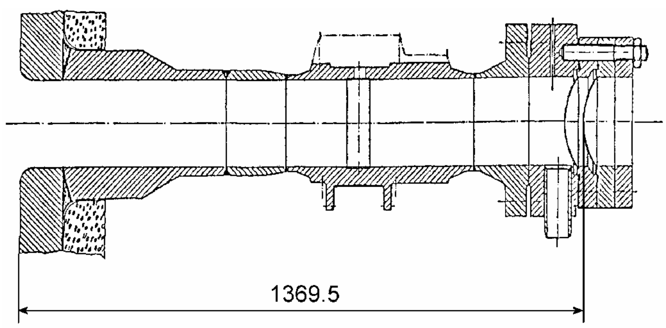

Figure 2. Break nozzle of the HDR reactor. [4]

Table 1. Main parameters of the HDR blowdown experiments and of the VVER-440 type PWR. [1,5]

Quantity	HDR	PWR
Pressure, MPa	11	12.5
$p_0 - p_{sat}$, MPa	5.5	7
$T_{core} - T_{downcomer}$, °C	0…50	30
Break diameter, m	0.2	0.5
Break opening time, ms	1…2	?
Core barrel length, m	7.6	8.1
Core barrel thickness, mm	23	50
Core barrel diameter, m	2.66	3.2
Maximum stress, MPa	100	230
Maximum displacement, mm	2	5

3. TWO-PHASE SYSTEM CODE CALCULATIONS

The system code APROS was used to evaluate the pressure boundary condition for the CFD calculation. The break nozzle was modeled with 45 nodes to be able to provide detailed time dependent pressure evolution during the first milliseconds after break opening (see Fig. 3). The opening time of the break was not known. Therefore, the external pressure boundary for APROS calculation was adjusted with the measured pressure from the HDR experiment to get correct pressure reduction rate in the outlet of the nozzle, i.e. break opening time of 1 ms. The pressure measurement in the experiment was located near the break disks. Because small nodes were used in the calculation, critical flow model was not used. The flow was calculated directly with the basic equations. The code calculated evaporation due to flashing and fluid acceleration inside the pipe. The node length in the break nozzle was 3 cm. Measured and calculated pressures at the nozzle outlet and calculated pressures with different nodalizations in the chosen location of CFD boundary condition are shown in Fig. 4. The pressure boundary condition for the CFD calculation was taken from a point inside the nozzle where there were not yet void to allow single phase calculation. Fig. 5 shows calculated break flows with 5, 15 and 45 nodes in the break nozzle.

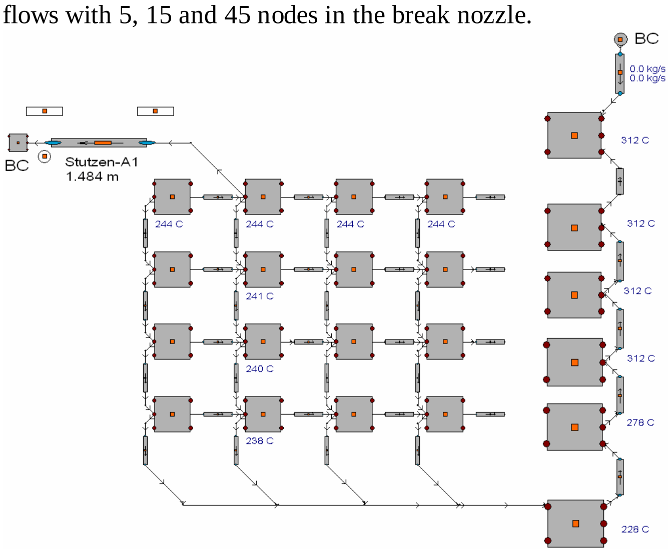

Figure 3. APROS model.

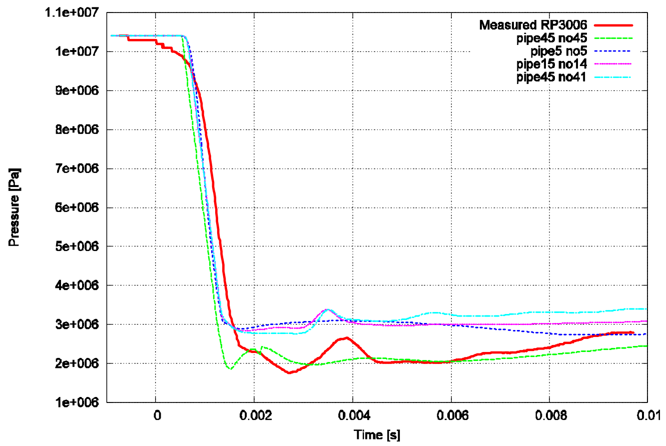

Figure 4. Measured and calculated pressures in the break nozzle. Boundary condition for the CFD model was taken from node 41 of the model with 45 nodes in the nozzle.

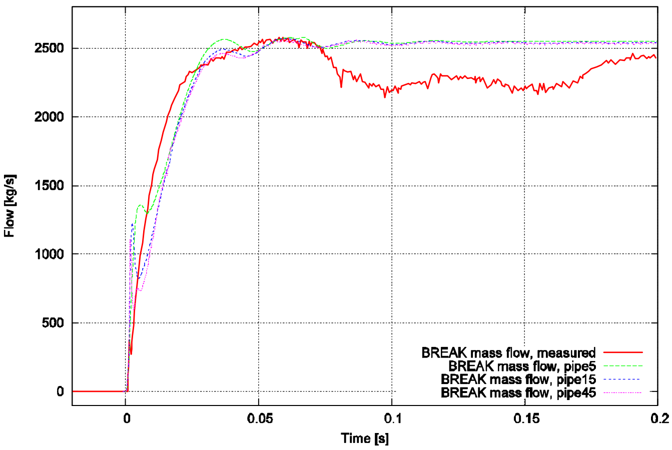

Figure 5. Measured and calculated blowdown flow.

4. NUMERICAL MODELS FOR FSI CALCULATIONS

Two-way coupled FSI simulations of the HDR experiment V32 were carried out by using the finite volume based CFD code Star-CD 3.24 and the Finite Element Method (FEM) based structural analysis code ABAQUS 6.6. FSI coupling was achieved by linking the fluid and structure codes with MpCCI 3.0.5. In addition to the coupled CFD-FEM simulations, acoustic-structural simulations of the experiment were carried out with ABAQUS.

4.1 CFD MODEL

Numerical mesh of the CFD model, shown in Fig. 6, had about 78000 hexahedral cells. In the downcomer, four cells were used in the radial direction. The used mesh resolution was considered sufficient for capturing the pressure wave propagation and overall flow phenomena according to earlier simulations of LBLOCA [1]. Wall friction, however, is not properly accounted for with this mesh.

Values ρ_0 = 758.5 kg/m^3, c = 1069 m/s and μ = 0.0001 Pa·s were used for reference density, speed of sound and molecular viscosity, respectively. These values were chosen according to average of the temperatures in the downcomer and upper core. Pressure calculated with the system code APROS (see Fig. 4) was used at end of the break nozzle as a pressure boundary condition.

The PISO pressure correction algorithm of Star-CD was used for numerical solution. The Crank-Nicholson method was used in the temporal discretization. In the spatial discretization, the Monotone Advection and Reconstruction Scheme (MARS) was used for velocities, central differencing for turbulence quantities and blended central-upwind differencing with blending factor 0.7 for density. Turbulence was modeled with the standard large Reynolds number k-ε model.

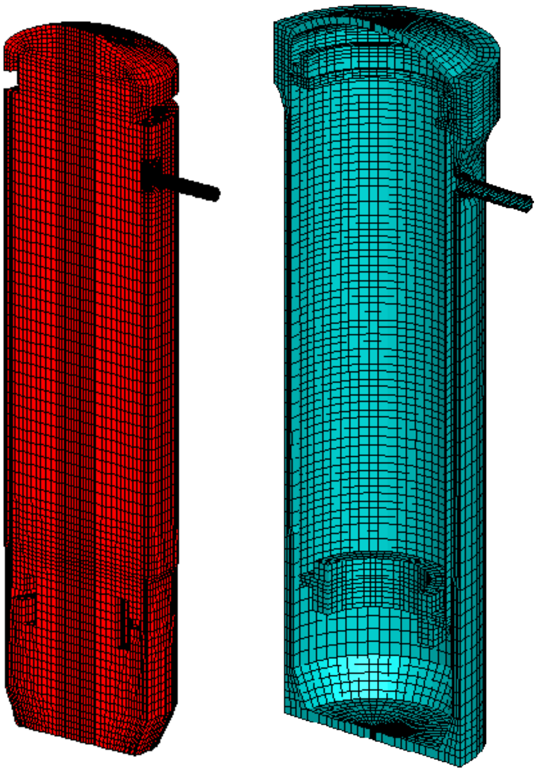

Figure 6. CFD (right) and structural (left) model meshes of the HDR reactor.

4.2 STRUCTURAL MODEL

A linear FE model of the reactor was created with about 15000 8-node hexahedral elements. Continuum shell elements, which have only displacement degrees of freedom (d.o.f.) but model shell behavior accurately, were mainly used. Conventional solid elements were used in few necessary regions. One layer of continuum shell elements was used in the core barrel wall and four layers in the reactor pressure vessel (RPV) wall. The mesh of the structural model is shown in Fig. 6.

Material properties E = 175 GPa, ν = 0.3 and ρ = 7900 kg/m^3 were used for the care barrel for elastic modulus, Poisson's ratio and density, respectively. For the RPV, values E = 190 GPa, ν = 0.3 and ρ = 7850 kg/m^3 were used [3]. A small amount of stiffness proportional Rayleigh damping was included in the RPV wall. Value $\beta = 6\times10^{-6}$ was used which results in 2 % of critical damping at frequency 1000 Hz. The maximum frequency of interest was estimated as 400 Hz.

4.3 COUPLING OF CFD AND STRUCTURAL MODELS

The Star-CD and ABAQUS codes were linked bi-directionally by using the external coupling software MpCCI. In this approach, the CFD and structural analysis codes run simultaneously and coupling information is exchanged during the simulation. Interpolation is used for transferring coupling quantities between the fluid and structure meshes.

In this work, the coupling quantities were fluid pressure and nodal coordinates of the structure. First, Star-CD sent the pressure load to ABAQUS. Then ABAQUS advanced one time step and sent the resulting deformations to Star-CD. After this, Star-CD advanced one time step and sent the new pressure load to ABAQUS. This cycle was repeated until the desired analysis time was reached. The time step size was 10 μs. The internal CFD mesh was smoothed with the mesh morpher of MpCCI. The used sequential solution strategy, where no iteration is performed on the coupled system inside a time step, results in explicit FSI coupling scheme.

4.4 ACOUSTIC-STRUCTURAL MODEL

In the coupled acoustic-structural FEM simulations, water was modeled as an acoustic medium and pressures and displacements were coupled on the fluid-structure interface. Contrary to the partitioned CFD-FEM approach, a monolithic solution strategy is used in the acoustic-structural simulation where a single system of equations contains both the fluid and structure d.o.f., see e.g. reference [7].

The same structural model and the same fluid mesh as in the CFD-FEM simulations were used. Linear 8-node acoustic elements were applied for the fluid. Interpolation for transferring quantities between the dissimilar meshes was performed internally by ABAQUS. Implicit direct time integration was employed with a time step 50 μs.

5. RESULTS OF FSI CALCULATIONS

Results of the CFD-FEM and acoustic-structural calculations and comparison with the experiment are presented in the following subsections 5.1 and 5.2. Time histories are presented from locations defined by a cylindrical coordinate system shown in Fig. 1, i.e. the break nozzle has coordinates z = 8850 mm and φ = 90º.

5.1 CFD-FEM CALCULATIONS

Figs. 7 and 8 show pressure distribution in the CFD model and deformations and stresses in the structural model at selected instants of time. The propagation of the first rarefaction wave into the reactor is seen in Fig. 7. The effective sound speed in the downcomer is significantly reduced due to flexibility of the structure. The pressure drop induces at first outward motion of the core barrel wall near the break. Later, swaying of the lower end of the core barrel can be seen.

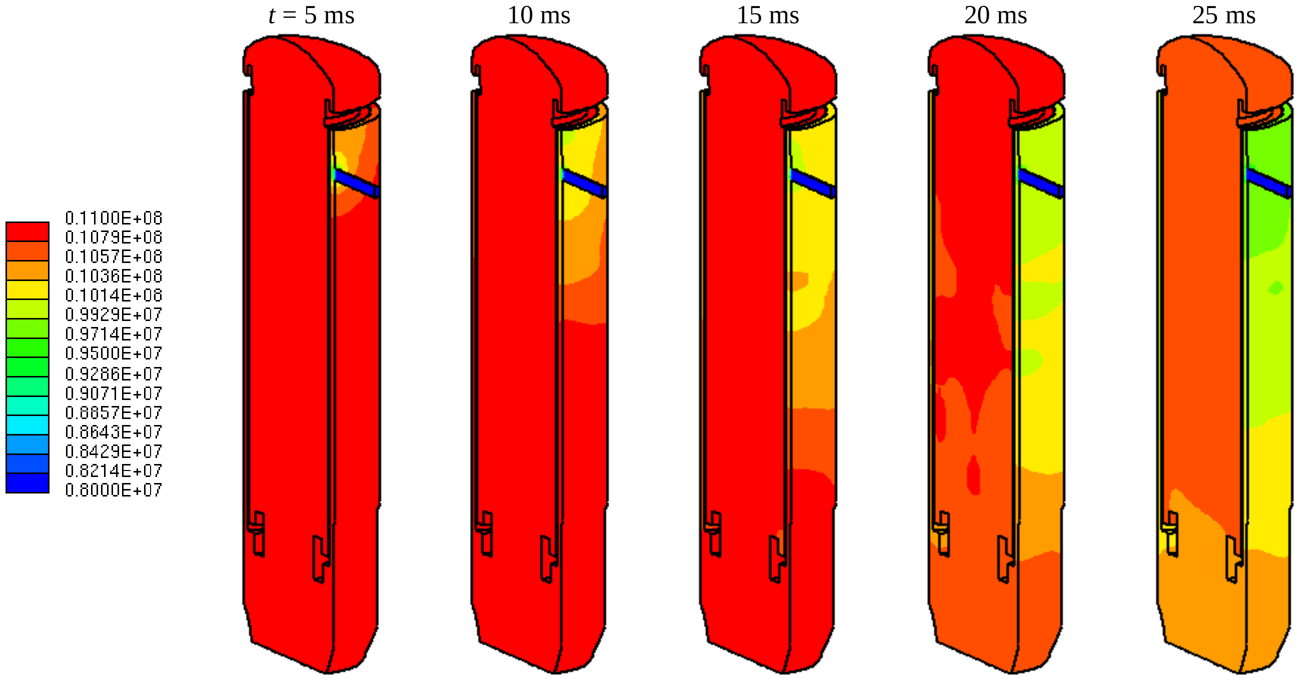

Figure 7. Pressure (Pa) in the CFD model at different instants of time.

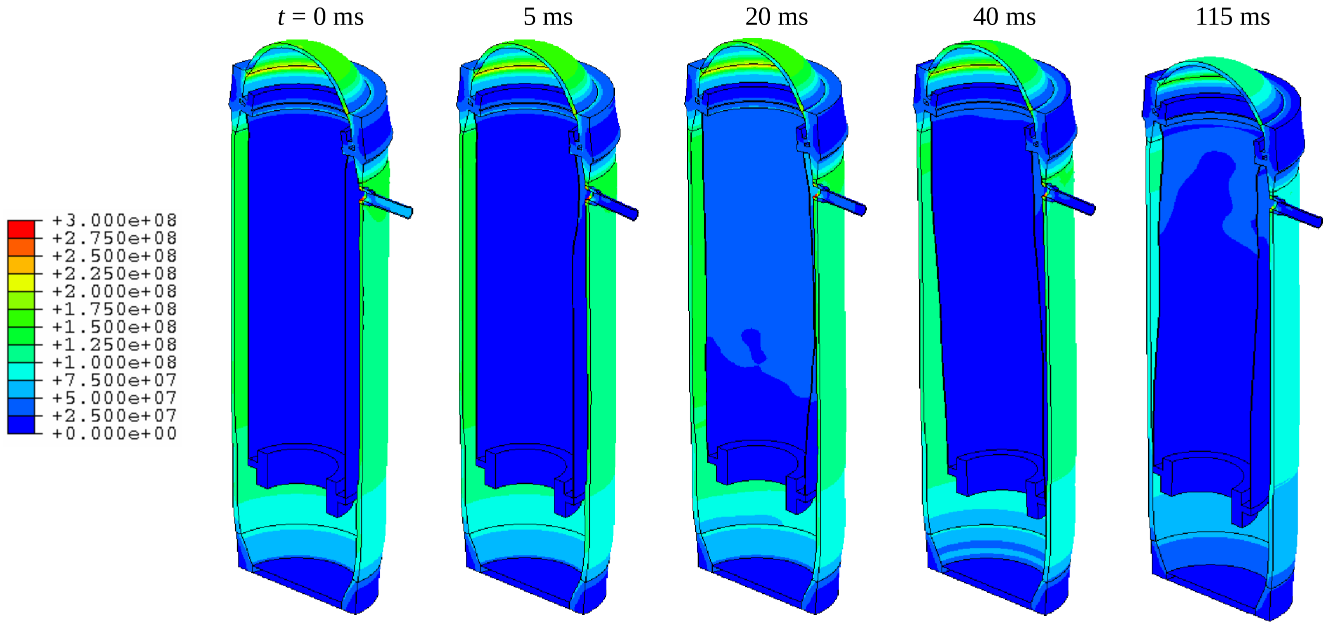

Figure 8. Von Mises stress (Pa) in the structural model at different instants of time. The deformation scale factor is 200.

Pressure in various locations in the downcomer and in the plenum of the reactor is presented in Fig. 9. As can be seen, agreement between the simulation and experiment is quite good during the first 120 ms. After this, steam production in the experiment is strong and the calculated pressures drop well below the measured values. The reason for the difference is that the effect of phase change is not included in the CFD calculation.

Fig. 10 compares pressure difference across the core barrel wall in the experiment and calculation. It is seen that the results agree quite well during the whole period, i.e. the pressure difference is fairly unaffected by the phase change.

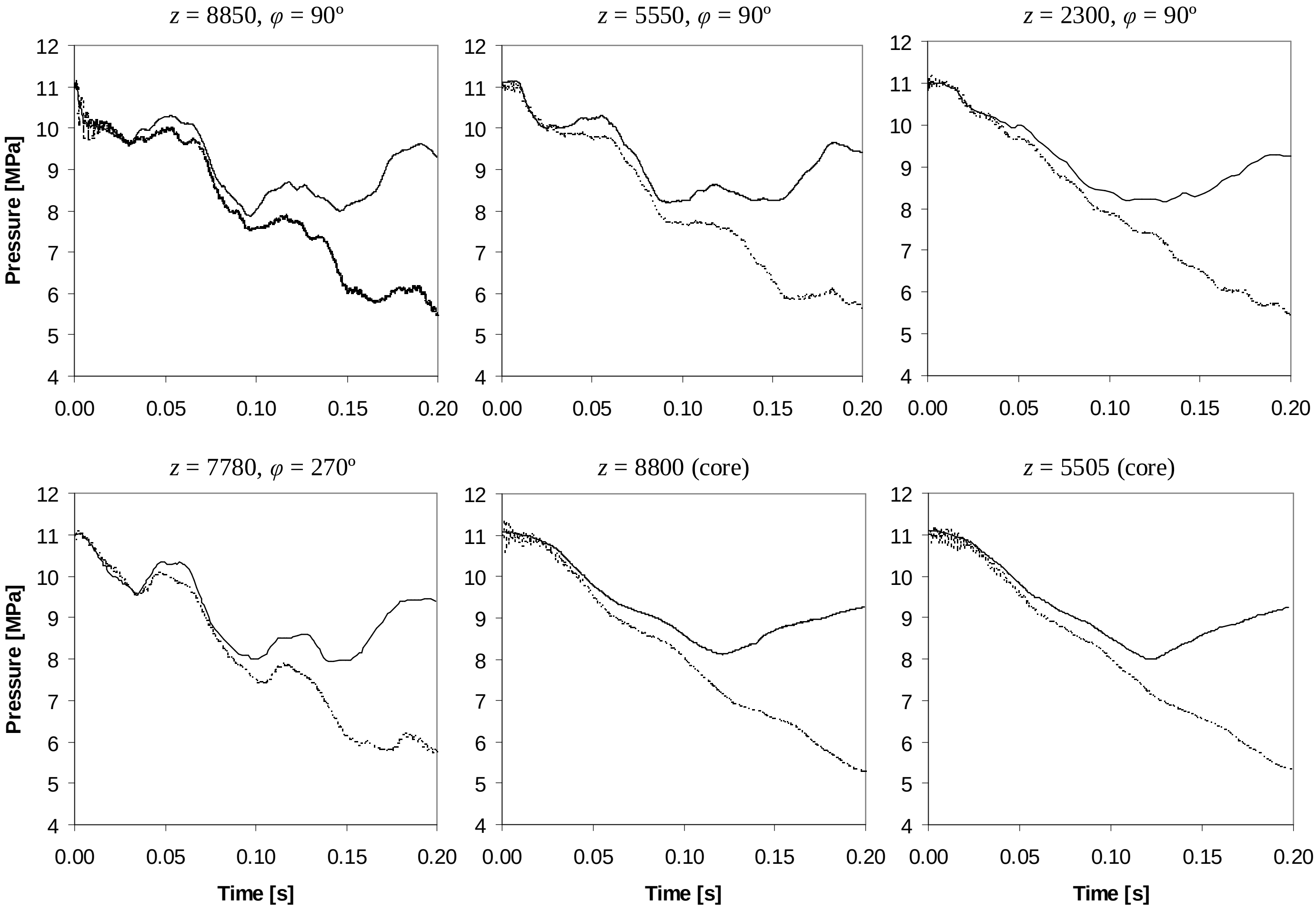

Figure 9. Pressure in the downcomer and in the core as a function of time. — experiment, --- CFD-FEM.

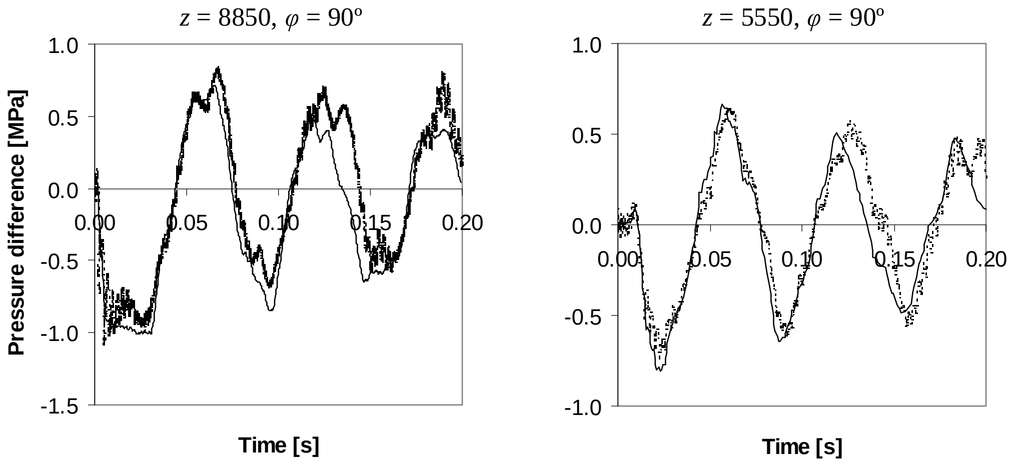

Figure 10. Pressure difference across the core barrel wall as a function of time. — experiment, --- CFD-FEM.

Relative radial displacement between the walls of the core barrel and RPV at various locations is shown in Fig. 11. Note that positive relative displacement means that the distance between the walls has decreased and vice versa. Agreement in the displacements is good during the first 0.1 seconds, although the displacements are slightly under-predicted by the calculation. In the late phase, larger differences are found. It can be noted from the results of the system code calculations in Fig. 4 that somewhat larger displacements would have been obtained by using the measured pressure as pressure boundary condition in the CFD model. The maximum displacements of the core barrel are only on the order of few millimeters, but they still have a significant effect on the fluid pressure owing to the predominantly acoustic nature of the problem.

Measured and calculated strains on the core barrel wall are compared in Figs. 12 and 13. The overall behavior of the calculated strains corresponds quite well with the experiment. The calculated values, however, oscillate strongly with frequency of about 380 Hz contrary to the experiment. The same oscillation is shown also in the calculated pressures and displacements especially near the break nozzle.

The oscillation with frequency of about 380 Hz, shown well in the calculated strains, is caused by an acoustic oscillation mode in the break nozzle. This frequency corresponds well with that of a sound wave traveling back and forth in the nozzle and was observed in a separate calculation with rigid walls and in the acoustic calculations as well. The reason why this oscillation is not shown in the experiment may be partly caused by boiling inside the nozzle (and outside it), which makes water “soft” in this region. In addition, the physical boundary of the problem is not fixed at the end of the break nozzle in reality. Andersson et al. [6] used the measured pressure signal as boundary condition in a similar single-phase simulation of HDR experiment V31.1 and their results showed only little oscillations. In this work, the calculated pressure boundary condition has a sharp drop which intensifies the oscillation (see Fig. 4).

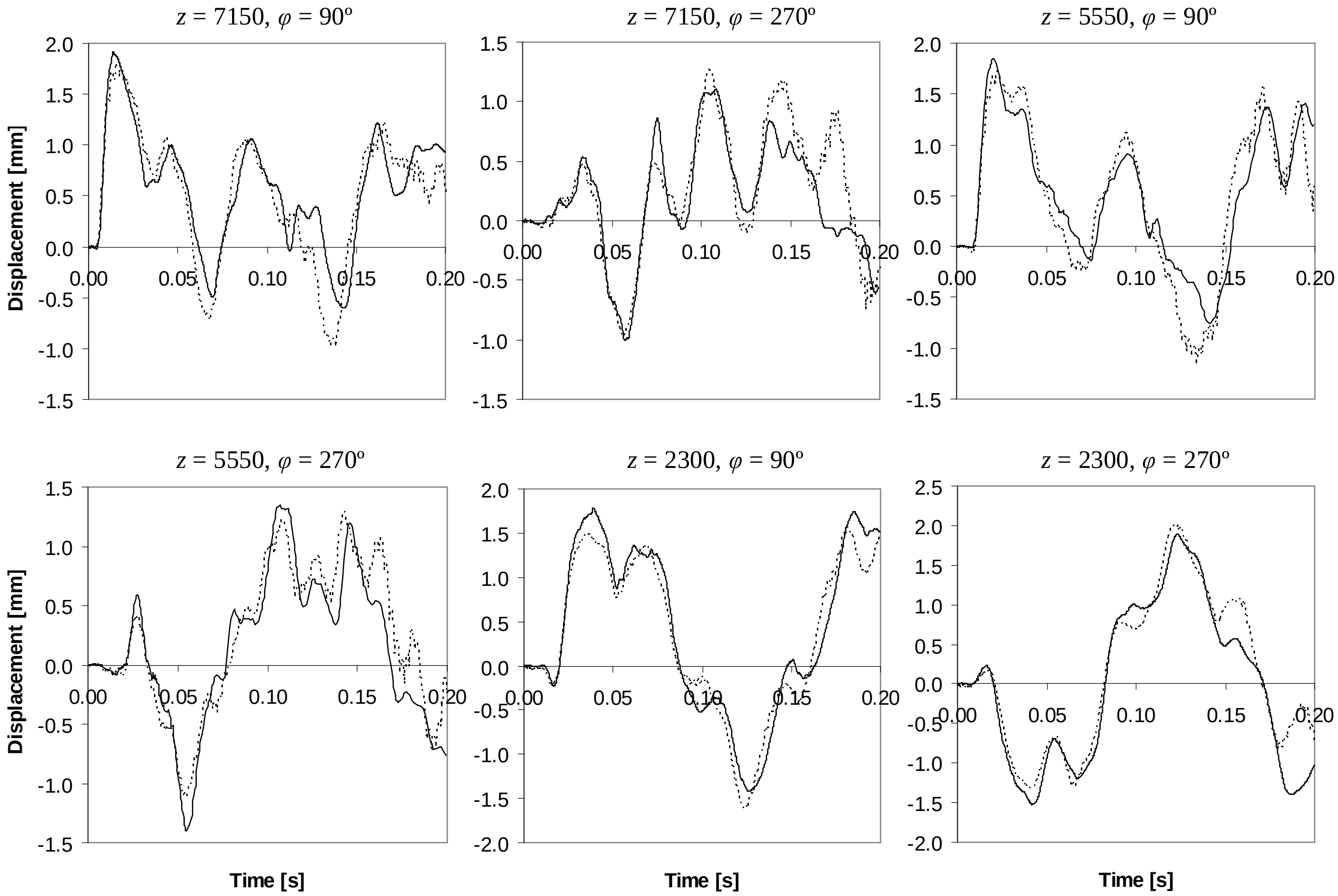

Figure 11. Relative radial displacement between the core barrel and RPV as a function of time. — experiment, --- CFD-FEM.

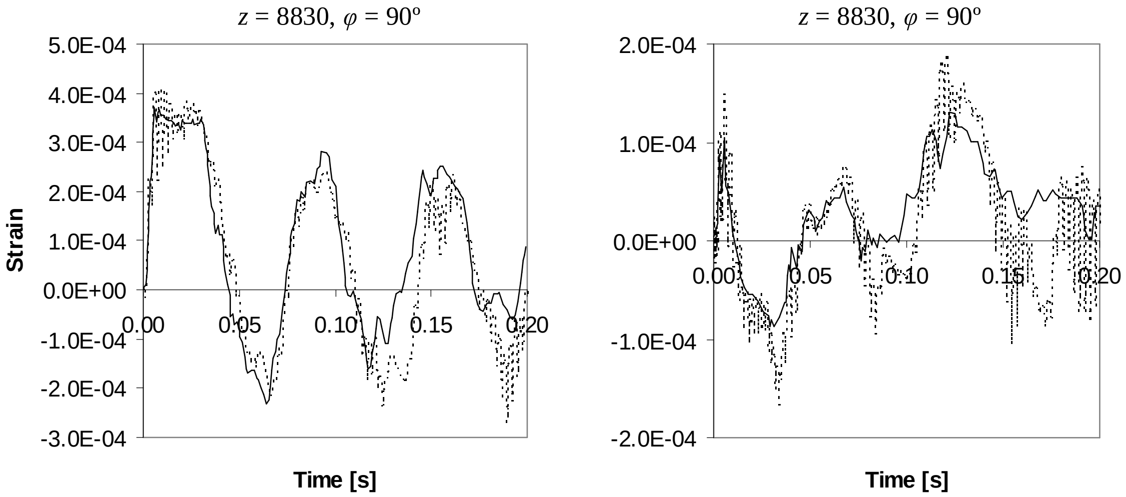

Figure 12. Hoop (left) and axial (right) strain on the outer surface of the core barrel wall as a function of time. — experiment, --- CFD-FEM.

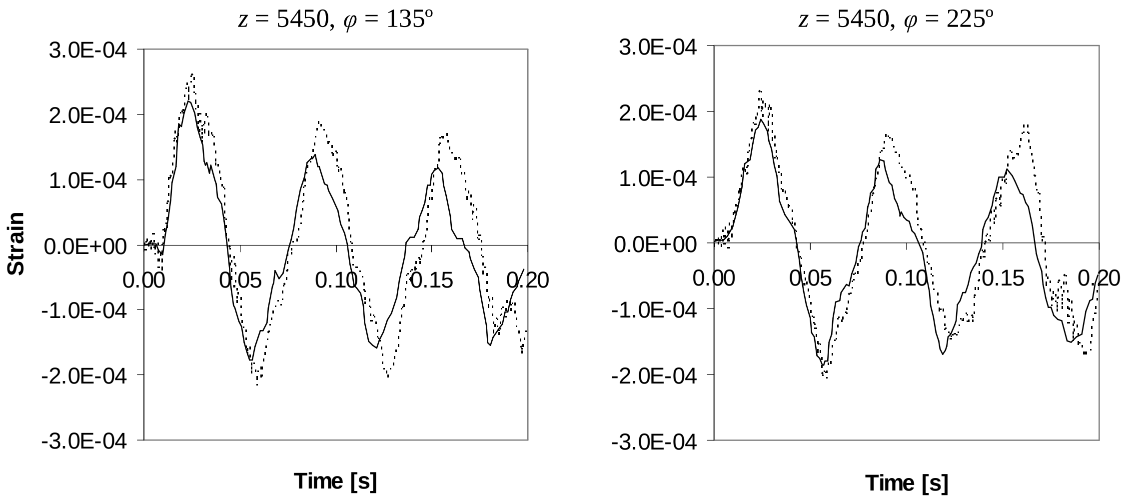

Figure 13. Hoop strain on the inner surface of the core barrel wall as a function of time. — experiment, --- CFD-FEM.

5.2 ACOUSTIC-STRUCTURAL CALCULATIONS

Pressures are compared in Fig. 14 from the first 100 ms of the transient. It is seen that the measured and calculated pressures deviate more than in the CFD-FEM calculation and that pressure loads on the core barrel are clearly over-predicted in the late phase of the simulation. This is due to the high flow velocity of water at the break location in the late phase, the effect of which is not included in the acoustic model.

Structural displacements and strains are shown in Figs. 15 and 16. It can be seen that the acoustic-structural model clearly over-predicts motion of the core barrel in the late phase, as was expected from the comparison of pressures. In the early phase, when the dynamic pressure is negligible, practically identical results are obtained with the acoustic-structural and CFD-FEM calculations and agreement with the experiment is quite good.

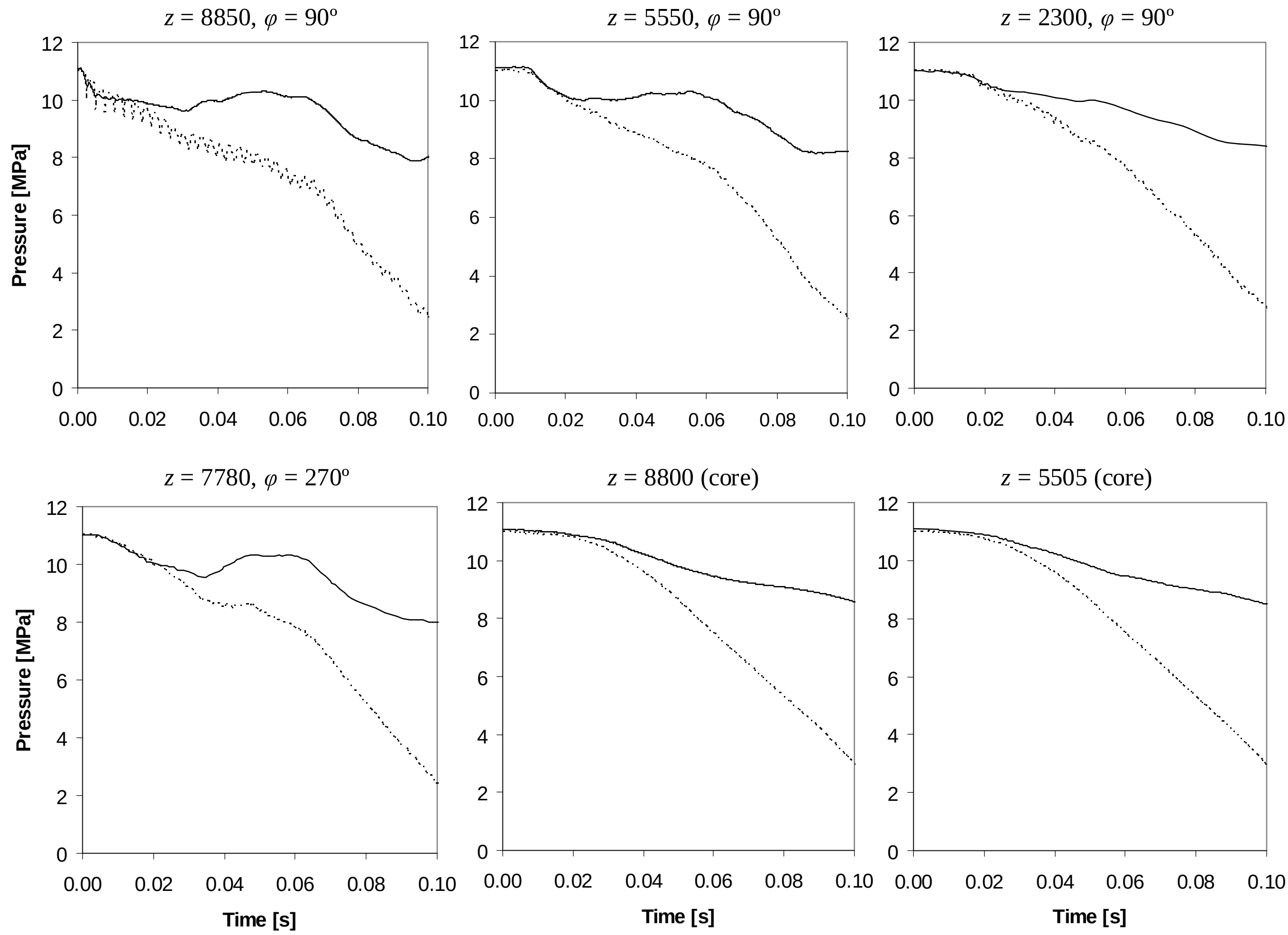

Figure 14. Pressure in the downcomer and in the core as a function of time. — experiment, --- Acoustic FEM.

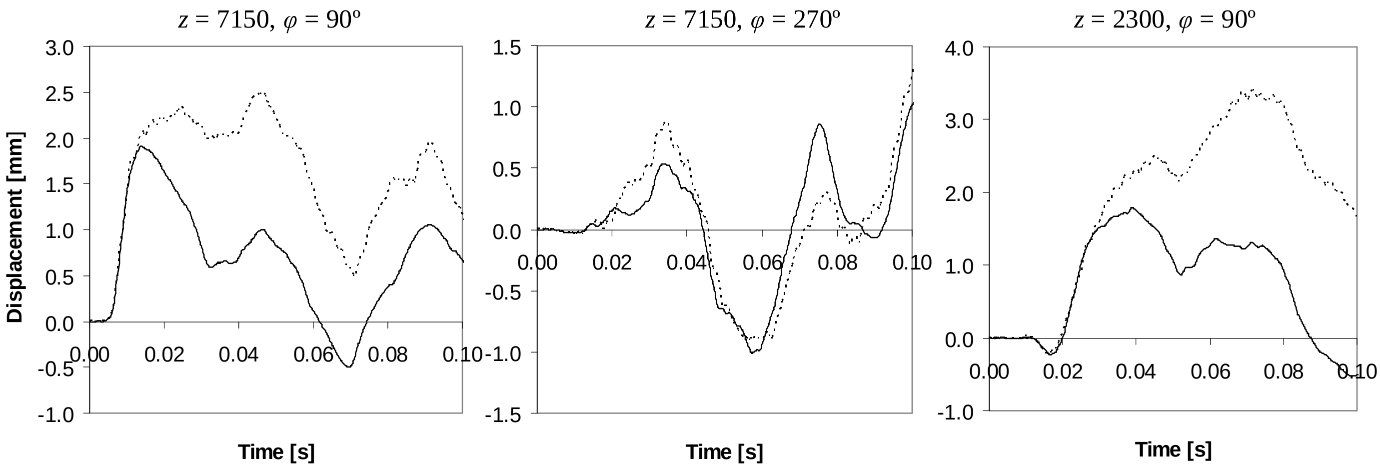

Figure 15. Relative radial displacement between the core barrel and RPV as a function of time. — experiment, --- acoustic FEM.

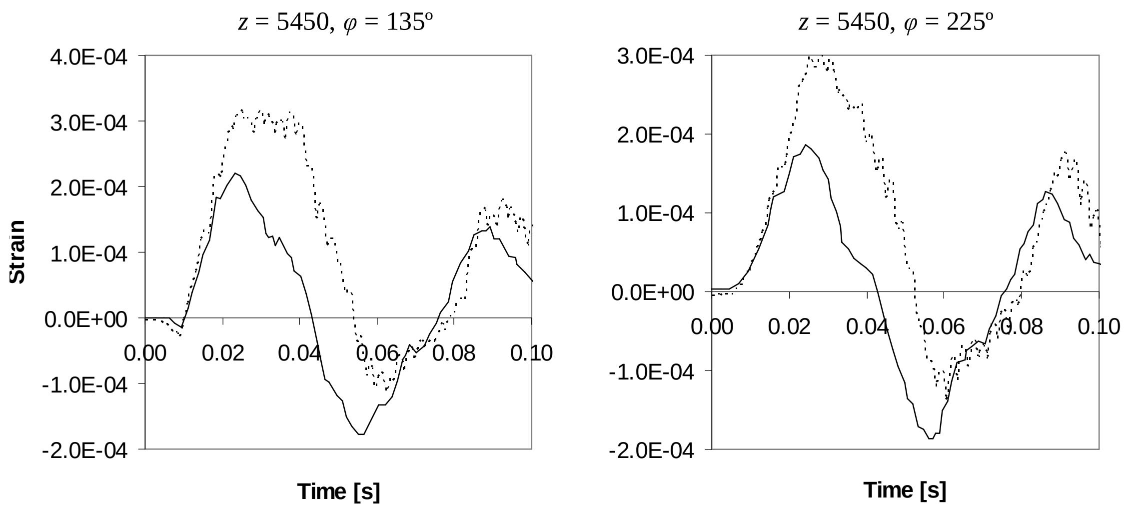

Figure 16. Hoop strain on the inner surface of the core barrel wall as a function of time. — experiment, --- acoustic FEM.

6. CONCLUSIONS

Results obtained with the coupled CFD-FEM method were in fairly good agreement with the experiment. Especially the overall structural behavior of the core barrel agreed well with the experiment. The results show that the most significant phenomena of the LBLOCA were well captured by the CFD-FEM calculation during the first 100 ms, when the largest structural effects occurred.

Results of the CFD-FEM and acoustic-structural calculations were practically identical in the early phase of the simulation. Maximum displacements and strains of the core barrel were, however, highly over-predicted by the acoustic calculation, owing to the effect of bulk flow of water which is not accounted for in the acoustic model. In earlier demonstration calculations of LBLOCA of a real PWR, a much better agreement in structural displacements was found between the two methods during the most significant period of the transient [1]. As the acoustic-structural method is considerably more efficient and more robust compared to the CFD-FEM method, it may still be considered as an alternative tool in future work.

Fairly coarse numerical meshes were sufficient for capturing the most important fluid transient and FSI phenomena. The single-phase assumption used in the FSI calculations seemed to be adequate during the phase when the largest structural effects occurred. However, probably an unrealistically short break opening time, i.e. about 2 ms, was considered in this and also in the earlier work [1]. More realistic estimation for the opening time in a real PWR could be obtained by considering also dynamics of the breaking pipe.

ACKNOWLEDGMENTS

The authors thank the National Technology Agency of Finland (Tekes) for financial support. This work is part of the SAFIR2010 programme (The Finnish Research Programme on Nuclear Power Plant Safety 2007 – 2010).

REFERENCES

[1] Lestinen, V., Toppila, T., Timperi, A., Pättikangas, T. and Hänninen, M., 2006. Determination of thermal-hydraulic loads on reactor internals in a DBA-situation. ASME Pressure Vessels and Piping Conference, Vancouver, Canada, 2006, PVP2006-ICPVT11-93456.

[2] Wolf, L., 1982. Experimental results of coupled fluid-structure interactions during blowdown of the HDR-vessel and comparisons with pre- and post-test predictions. Nuclear Engineering and Design **70**, 269–308 (1982).

[3] Wolf, L., 1981. Design report for the HDR-RPV-I blowdown experiments V31.2, V32, V33 and V34 with specifications for the pretest computation. HDR Safety Program, Report No. 3.243/81, Kernforschungszentrum Karlsruhe.

[4] Wolf, L., Schall M., and Bader, Hj., 1983. Untersuchungen von RDB-Einbauten bei Bruch einer Reaktorkühlmittelleitung, HDR Sicherheitsprogramm, Technischer Fachbericht 29-82, Kernforschungszentrum Karlsruhe.

[5] Schall M., 1984. Untersuchungen von RDB-Einbauten bei Bruch einer Reaktorkühlmittelleitung, HDR Sicherheitsprogramm, Technischer Fachbericht 48-84, Kernforschungszentrum Karlsruhe.

[6] Andersson L., Andersson P., Lundwall J., Sundqvist J. and Veber, P., 2002. Numerical simulation of the HDR blowdown experiment V31.1 at Karlsruhe. ASME Pressure Vessels and Piping Conference, 2002, PVP2002-1128, **435**.

[7] Cook, R., Malkus, D., Plesha, M. and Witt, R., 2002. Concepts and applications of finite element analysis. Wiley & Sons, Inc.

