"Drop-Test" FSI simulation with Abaqus and FlowVision based on the direct 2-way coupling approach

A. Aksenov [1)], D.Korenev [1)], A. Shyshaeva[1)], D. Vucinic [2)],
Z. Mravak [3)]

[1)] TESIS Ltd, Russia [2)] Vrije Universiteite Brussel, Belgium, [3)] Bureau VERITAS, France.

Abstract:

The paper presents a numerical simulation of the drop test in a still water for the multi-component box structure. The complexity of the problem is in the strong fluid-structure interaction (FSI) between the box and the water free surface. The numerical simulation of the drop test is performed with two software tools: Abaqus and FlowVision through the direct coupling interface, which manipulates, on the Abaqus side the Lagrangian finite-element mesh and on the FlowVision side the Eulerian finite-volume mesh with subgrid geometry resolution. The novel approach is that there are no auxilliary structure models (or 3rd party software) integrated in the applied software solution: the finite-element mesh is defined from the Cartesian CFD finite-volume mesh and all the relationships between the CFD mesh cells and the outside FE faces are fully preserved. Each mesh node displacement is directly transferred between FlowVision and Abaqus, thus avoiding any additional interpolation.

Keywords: Shipbuilding, CFD Coupling, Impact, Fluid-Structure Interaction, Free Surface flow, Slamming

1. Introduction

The analysis of the drop of containers in water have significant interest because of its presence in different application domains, e.g. military (landing loads in water for supplying troops) and saving dangerous cargo during ship transportation operations (Faltinsen , 200). The container must stand against the impact and to protect its content from a possible damage. The simulation of the container drop testcase when container falls on the rigid surface can be solved with finite-element (FE) analysis codes; in our approach we demonstrate the application of the Abaqus/Explicit FE code.

The problem of the impact of a container with the water surface represents a complex problem, which requires simultaneous calculations of a water flow around the container and a stress analysis of the container under investigation.

Results of a container impact with the water surface strongly depends on the following factors – the angle between bottom side of a container and the still water surface, its speed and the impacted surface geometry.

Simplifying the impact and interaction between the structure and water, the simulation can be split into two parts – the pure hydrodynamic loading on a solid structure and the calculation of a stress in container structure yield under the defined loadings. Such simplification could be applied for the ceses with a low speed, where the interaction is not so important.

At high speeds and/or small angles a high pressure is generated between the bottom of a container and the water surface. The application of the load force results in the container structure deformation or in its internal content shift. The strong interaction between the surface deformation and the fluid flow is present, and in such casewe suggest that the simulation is performed using a 2-way coupling approach between the FE analysis code and the CFD code, in order to take into account the change of a container shape and its center mass position during the drop while submerging into the water.

In this paper, the slamming container testcase is solved as the strong fluid-structure interaction (FSI) problem. The container has a complex multi-component structure and it falls into a still water. FSI problem is approched by a 2-way direct coupling between FE code ABAQUS and finite-volume CFD code FlowVision. ABAQUS simulates the motion and the stress of the modeled container, while FlowVision calculates the water loads, which are acting on the container surface. Different mesh types are applied for the container model, consisting of the solid structure on one side and the fluid on its another side. The container mesh geometry is modeled by mixed tetrahedral and hexagonal Lagrangian finite elements; the fluid domain is modeled by Eulerian finite-volume cells supporting sub-grid geometry resolution. The important issue is that there is no any other intermediate structure model (or 3rd party software) applied in the presented simulation: the FE mesh is defined from the Cartesian CFD finite-volume mesh and all the dependencies between CFD mesh cells and outside faces of the finite elements are fully controlled. Node displacements are transferred directly between FlowVision and Abaqus avoiding any additional interpolation.

The container drop test model is used for analyzing different initial angles between container bottom and water surface. At small or zero angles, the physics of a container impact with the water surface is rather complex, as it has to take into account the interaction of an air flow in the quite narrow gap between the container bottom and the the water surface, where in addition the water compressibility is modeled. It has to be noted that the air between the container bottom and the water surface could have a cushioning effect, which is not taken into account in this paper and we plan to research it in the near future.

It is important to mention that the compressibility of the water is taken into account, because ofignoring water compressibility results in unphysical simulation results. This *ex facte* strange resume (this problem is characterized with very low Mach number of the water flow, which speed is about 10 m/s, when compared with 1.5 km/s of sound speed in the water) is discussed further on.

As mentioned previously, we have made the simulations of 3 drop test cases. We show the importance of taking into account the water compressibility in order to simulate the impact of the container box-structure with the water. The water pressure on the bottom part of the container and the stress inside the container are calculated as the functions of time. The methodology of the drop test simulation is discribed and it is expected to be applied for the validation of the real drop test simulations with existing experiments.

2. Statement of the problem

The multi-component box structure is dropped in a still water. The box consists of three major components: the steel housing, the foam inner damper and the bottom flat steel membrane, see Figure 1, while the container dimensions are shown in Figure 2. The modeled container falls in the water from the specified height of 2 and 3 meters. The initial impact angle is 4˚ and 2˚, as shown in Figure 2.
The strain values on the elements of the inner container structure (Figure 3, a) and the pressure values on the bottom of a membrane (Figure 3, b) are recorded during the wet drop simulation.

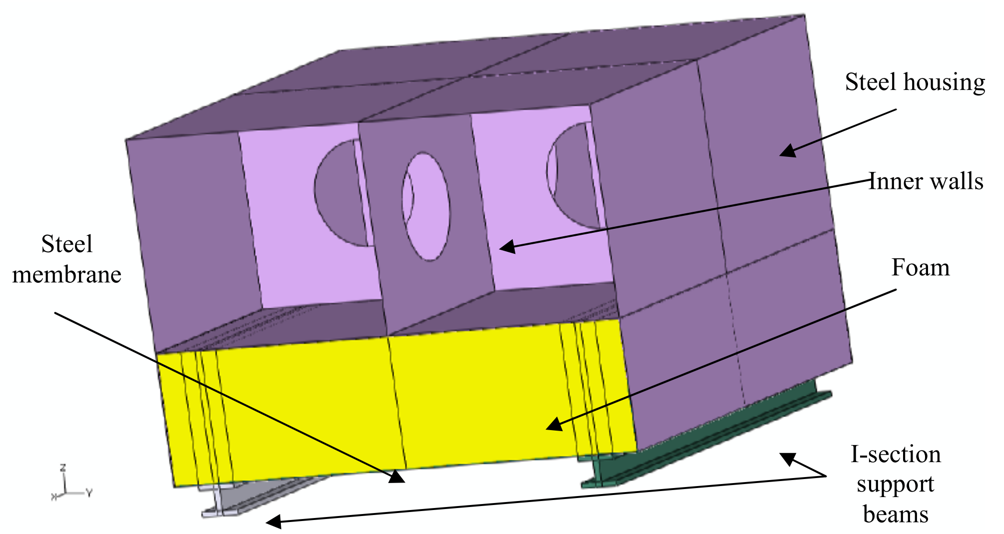


Figure 1: Components of testing drop-test structure

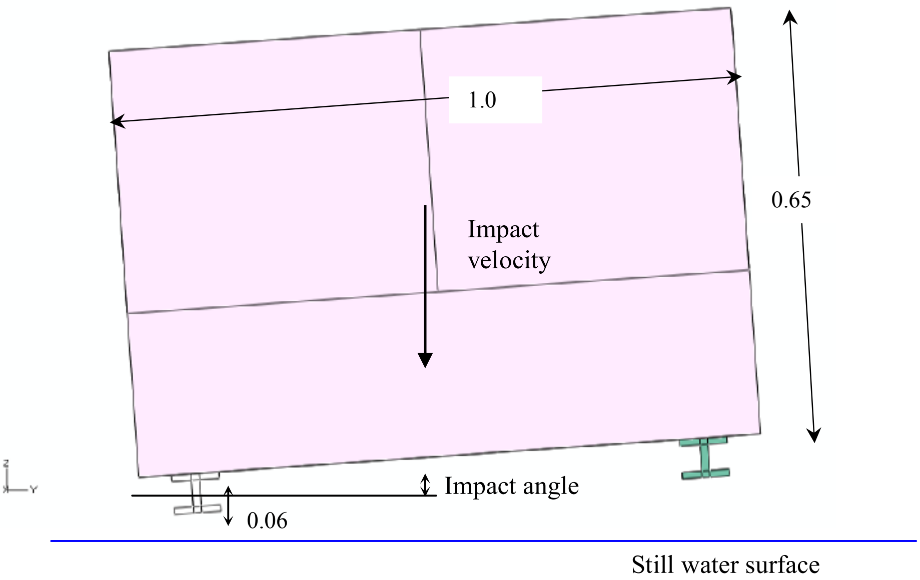


Figure 2: Drop test conditions

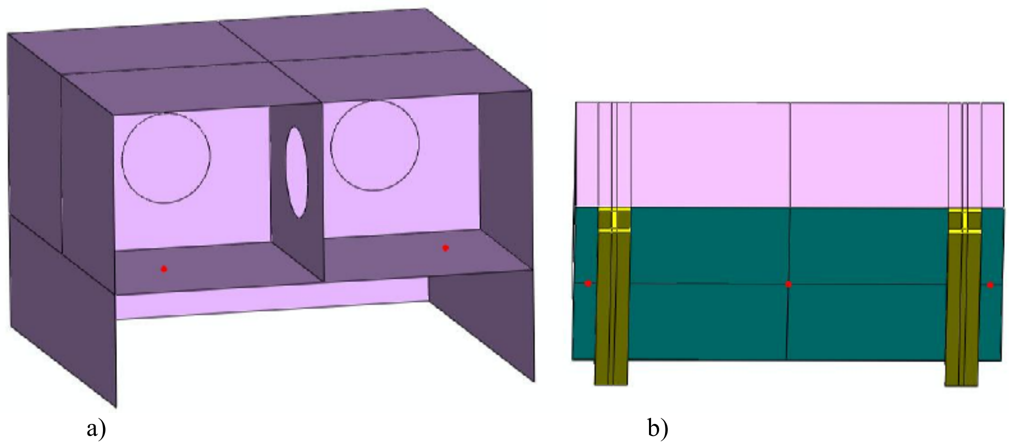


Figure 3: Location of points for values time history: a) on inner container structure (for strain values) b) on impact surface (for pressure values)

3. Simulation Model Definition

3.1 FEM Model

The container with prescribed internal structure is defined in ABAQUS and modeled by the FE mesh. The defined element types are arbitrary and do not effect the coupling between ABAQUS and FlowVision. ABAQUS/Explicit solver is used for calculations. The FE model consists of solid elements and 3D shell elements. The whole FEM mesh has about 42000 elements.

3.1.1 Properties and Materials

Part	Elements type	Section type	ρ, kg/m^3	E, Pa	μ	Notes
Drop Device walls and compartments	S4R, S3R	Shell, Homogeneous	7850	~2000×10^8	0.3	
Bottom membrane	S4R, S3R	Shell, Homogeneous	7850	~2000×10^8	0.3	
Foam	C3D8R	Solid, Homogeneous	110	En=130MPa Es=130MPa Et=60MPa, G_{ns}=14MPa, G_{nt}=14MPa, G_{st}=14MPa,	μ_{ns}=0.25, μ_{nt}=0.2, μ_{st}=0.2,	Modeled as orthotropic material
Wrapper	SFM3D4R,	Surface	No mass	No stiffness	-	Auxiliary part without mass or stiffness

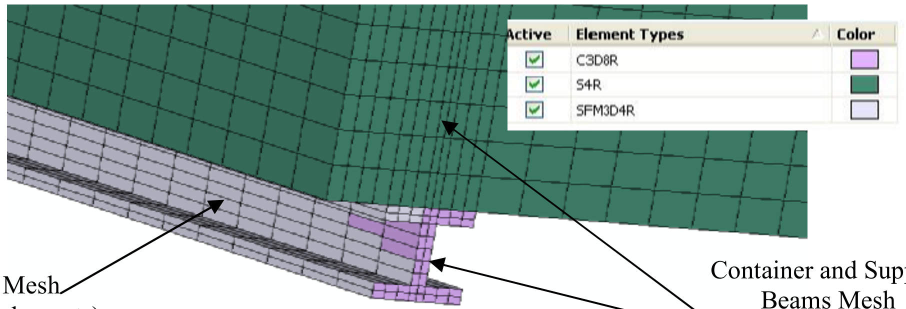
Table 1: Properties and materials used in FEM

3.1.2 Loads and Boundary Conditions

The gravity force is taken into consideration and the initial speed of the container is verticaly oriented with values of 7.28 m/s for the first drop test and 5.94 m/s for the second one.

3.1.3 Setup of FEM for CO-SIMMULATION

The interaction surface for the FSI simulation is defined as part of the Instance mesh. The container has complex exterior which can’t be used in co-simulation technique with FlowVision. First, external face elements of the container must be modeled as continues surface mesh without “holes”. To fulfill such condition, all the partitions, which come in a contact with the water surface, are merged together as one **Mesh Part**. In addition, an offset shell layer is created on the external side of the mesh by applying the command **Mesh→Edit Mesh→Mesh→Offset (create shell layers)** with zero offset; the **Surface** property is assigned to these offset elements (Figure 5). The wrapper mesh is saved in a separate file, as it represent an important input to FlowVision, defining the interface region for the co-simulation.



Wrapper Mesh
(Surface elements)

Container and Support
Beams Mesh
(3D and 2D)

Figure 4: Wrapper mesh defined for the container surface

3.1.4 Parameters of co-simulation

The co-simulation parameters in the step module of the input file are defined as follows:

```
*CO-SIMULATION, PROGRAM=DIRECT, NAME=FV_TEST, CONTROLS=COSIM_CONTROLS
**
*CO-SIMULATION REGION, IMPORT
test, CF
**
*CO-SIMULATION REGION, EXPORT
test, COORD
**
*CO-SIMULATION CONTROLS, NAME=COSIM_CONTROLS, TIME INCREMENTATION=SUBCYCLE,
TIME MARKS=YES
```

It is important to note that the concentrated force at each node is imported in ABAQUS and the nodes coordinates are exported from ABAQUS. ABAQUS allows sub-cycling during the coupling step.

3.2 CFD model

3.2.1 Computational domain

The CFD computational domain has the size of 5x6x6 m, it is 5 times larger than the container computational domain (Figure 5), with the modeled water depth of 3.65m. The large distance between outside walls of the computational domain is selected to prevent the influence of the reflected water waves, which originates from the external domain boundary during the container impact with the water surface.

The container geometry is imported in FlowVision using the “moving body” tool as the ABAQUS FE mesh. During the import, FlowVision automatically defines the external surface mesh in order to take into account the outside faces of the FE mesh. The FlowVision mesh has 130 000 cells and it is locally refined to the level 1 in the impact region (figure 6).

The FlowVision time step is automatically defined and it is close to, or equal 5x10-4s. The motion of the water is modeled with Navier-Stokes equations for incompressible fluid. Turbulence is taken into account by applying the k-ε turbulence model.

An advanced Volume of Fluid (VOF method is used to track the water free surface, which is specified with the distilled water parameters (density, viscosity). Initially, the container is just above the water surface. The initial vertical speed of the container is 6 m/s. The container is affected by the gravity force. The simulation of the container motion is done by ABAQUS/Explicit, taking into account the gravity force, the hydrodynamic forces and the resulting deformations..

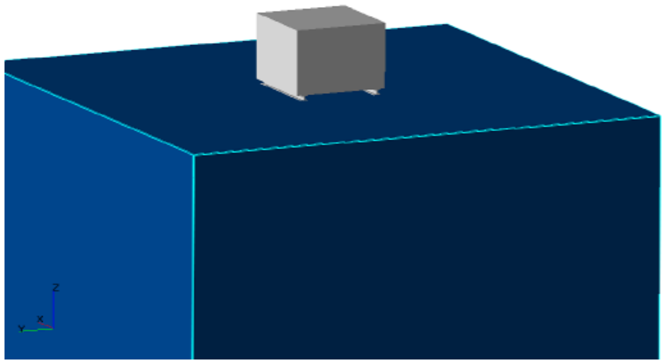


Figure 5: The CFD computational domain

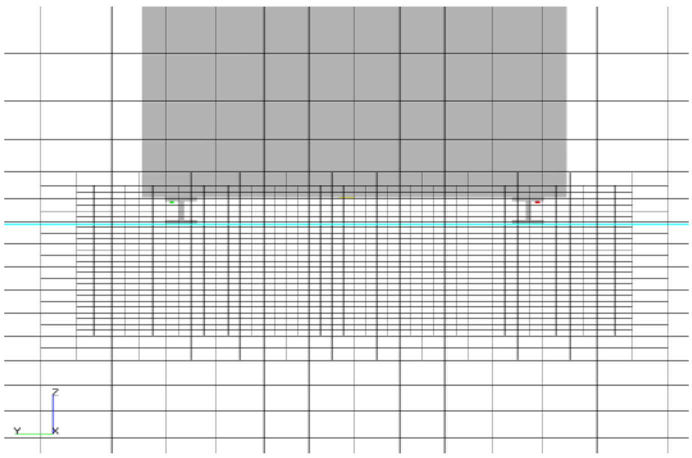


Figure 6: The CFD mesh with local adaptation near impact region

3.2.2 Set up of FlowVision and Abaqus co-simulation

The detail description on technical issues of the direct coupling interface between Abaqus FE mesh and FlowVision finite-volume mesh can be found in our previous works (Aksenov et al.,

2004, 2006). In the direct coupling procedure FlowVision is the master application and Abaqus works in a slave mode.
In order to create a FlowVision project with co-simulation a user is importing the container geometry inside the CFD computational domain. A mesh around the container has to match the both FE and CFD meshs, and it is built automatically created without any user operations. When FlowVision starts a simulation, it remotely invokes Abaqus too. For using Abaqus through the direct coupling interface (DCI) a connection is established via sockets. After the "handshaking" procedure through DCI, FlowVision sends to Abaqus the initial zero loadings, on which basis Abaqus makes a first time increment and sends back to FlowVision the newly computed container node coordinates..

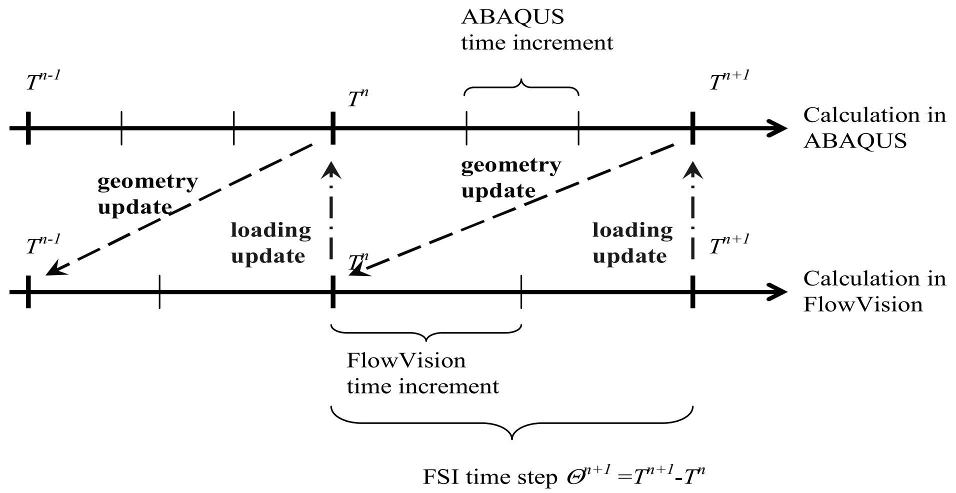


Figure 6.1: FSI time step for FlowVision and Abaqus

Explicit coupling procedure is applied for the time stepping algorithm. The exchanged data is consisting of forces computed in the nodes of the FE mesh, which comes from FlowVision and the node displacement exchanged data coming from Abaqus. The exchange time period (FSI time step) is based on the value of the CFD explicit time step (min h/V over all CFD domains:h- cell size, V – fluid speed). As practice shows, for the fast simulation processes, like the analyzed impact container with water is, the best choice of FSI time step is 1 or 2 CFD explicit steps.
The FSI time step is started by Abaqus and uses loadings from the previous FSI time step, as shown in figure 6.1. FlowVision receives new node coordinates and calculates new loadings.

3.3 Performance of co-simulation

All results are received using 1-processor machine (Intel Core, 2.4 GHz). FlowVision is runing on the Windows XP machine and Abaqus on the Linux machine. The used office network operates with 100 Mb/s which determines the data exchange speed between the involved computer resources. The exchange domain contains about 7500 nodes applied for the analyzed problem and the related simulation is taking approximately 2 hours on these specified machines. The simulation wall time for Abaqus and FlowVision is approximately 1hour, and the simulation process wall time for the data exchange is negligible and don't affect the simulation process wall time.

4. Results

The simulation of the container drop is performed with 3 different angles (0^o, 2^o and 4^o) between the container bottom part and the still water surface. Ex facte, this problem is appropriate to be solved applying the incompressible fluid model for the water motion. Indeed, the Mach number in this process is far too small (characteristic water speed is about 10 m/s, sound speed in the water is about 1400 m/s), and any water pressure value in the contact area between the container and the water is calculated. What happens is that after the impact, the container accelerates the water mass for about $\rho S^{3/2}$, where ρ is water density and S is container bottom area. The water is accelerated by pressure $P = \rho S^{1/2} V/\tau$, where τ is time of the impact. As the impact in the still water is an instant process, τ->0 and P->∞, which results in a non-physical state. In order to avoid such non-physical outcome and get the physically correct behavior, a compressibility of the water is added to the applied Navier-Stokes equations, see Korobkin, 1997, where the pressure is limited by value $P = K\rho c^2/2$ (c is the sound speed in the water, coefficient K is about 1). During the simulated impact, the container generates acoustic pressure waves within the water domain. In Figure 7 we show the pressure wave in water after 0.001 sec after the impact of the container bottom area with the water surface.

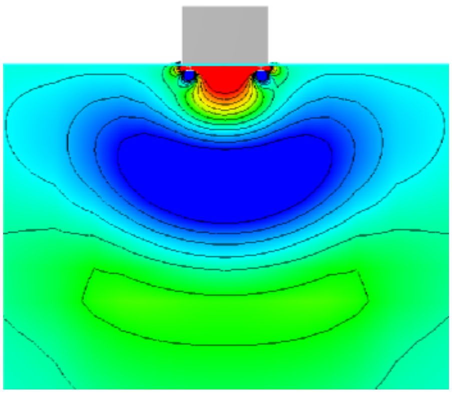


Figure 7: Acoustic wave after impact of the container with water, 0 degree inclination

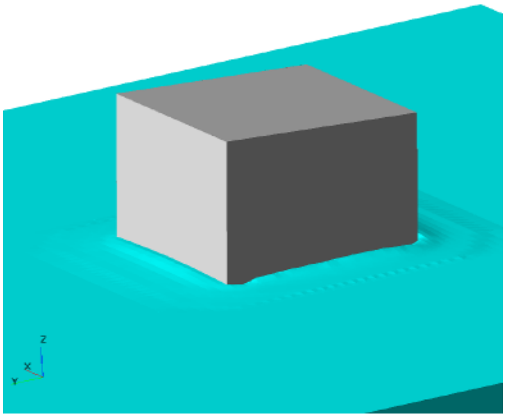


Figure 8: Free surface distribution after impact, 4 degree inclination of container

The free surface distribution at time0.015 s after the impact (4 degree of inclination) is shown in Figure 8. Immediately after the impact, the water is extruded from the space under the container and the near walls wave of the container is generated.

The water flow, along the container bottom area, just after the impact is shown in Figure 9. At the zero angles the water speed over the whole container bottom is small. It means that the huge pressure will cover almost the whole container bottom area. The container supporting beams are preventing the water flow to be oriented via borders where they are located, and the resulting water flows are mainly aligned via other borders. Small inclination of the container before the impact is dramatically changing the flow field picture. When the container is inclined, one of these container beams (right side, in Figure 9) penetrates first into the water, while the second is still outside of the water. The water could flow in all directions out from this first beam. We can observe that a lot of the container bottom area is flowed by the water at high speed, which results in decreasing the impact force.

The pressure distribution is shown in Figure 10, where we can observe the difference of the pressure distribution over the container bottom area. There are two high pressure regions at the zero degree inclination of the container that are shifted to the border of the bottom area. At non-zero angles inclination a high pressure region is located at the intersection line between the water surface and impacted surface. The time dependence of a pressure at the center of the container bottom is shown in Figure 12. We can observe that the pressure dependence has two peaks for the non-zero inclination angle at 0.015 sec and 0.016 sec, located at the line crossing the bottom center with still water surface. First peak is higher than the second one. This second peak is explained by the hit of the water wave to the second support beam of the container that is the latest part entering the water. In addition, the stroke of the wave towards the beams generates a strong acoustic wave that results in the increased pressure over the container bottom area. The impacted surface, with some appendices, in general has a completely different fluid flow structure during the impact, when compared with the flat surface test case. In general, such perturbation results with the smaller peak loads on the impact surface, phenomena known as the cushioning effect.

The bottom membrane and the support beams are affected by the strong pressure loadings. As results, we have large strains and deformations, generated during impact. The magnified bottom part of the container and the beam deformation (with scalar factor 100) is shown in Figure 12. The beam bending is visible inside the container with maximum deformation at the center of the beam. We found that the maximum principal stress exists near junction of the container bottom membrane and container walls, as shown in Figure 13. The peak of the strain in the membrane-wall junction is (for 2 degree container inclination), as follows:

$$\sigma_{max\_pr} = 2.2\times10^7 \text{ Pa}$$

The yield stress for the steel membrane is

$$\sigma_y = 1.8\times10^8 \text{ Pa}$$

The Margin of Safety (MS) is evaluated by expression

$$MS = \sigma_y / \sigma_{max\_pr} - 1 = 7.29 >> 0$$

In our case the Margin of Safety is much higher then 1, which means that the container design is far from the optimal and must be enforced in order to gurantee the safety conditions of the container contents.

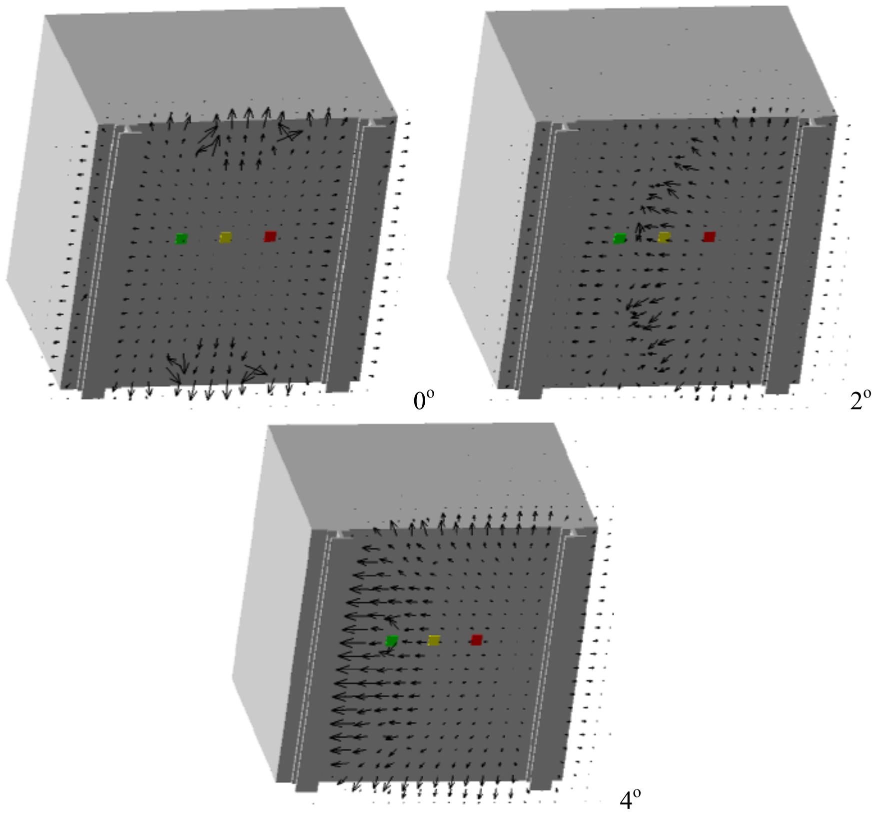


Figure 9: Velocity distribution in plane near bottom, t=0.015s after start motion.

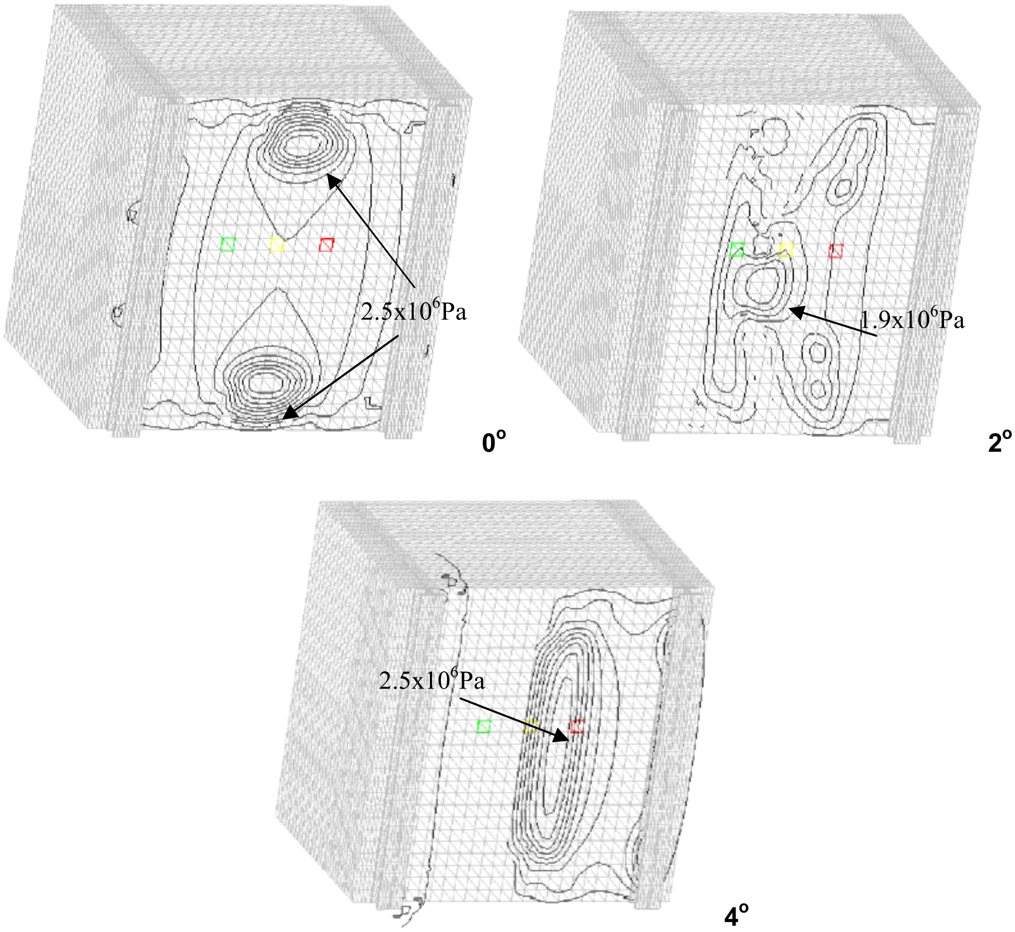


Figure 10: Pressure distribution in plane near bottom, t=0.015s after start motion, isolines step = 3x10^5 Pa

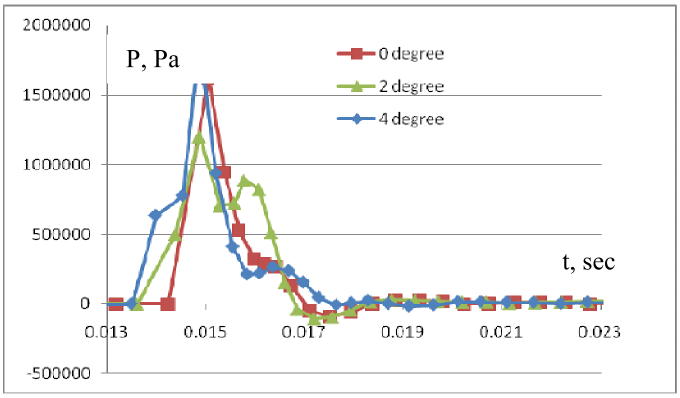


Figure 11: Dependence pressure on time at area of pressure sensor at the center of a bottom

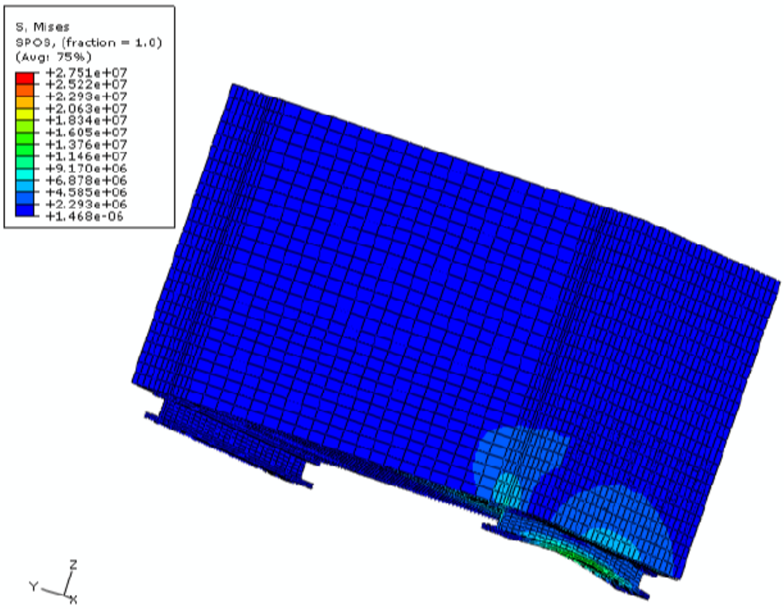


Figure 12: Deformation of support beams at first moments of container collision with water (deformation scalar factor 100)

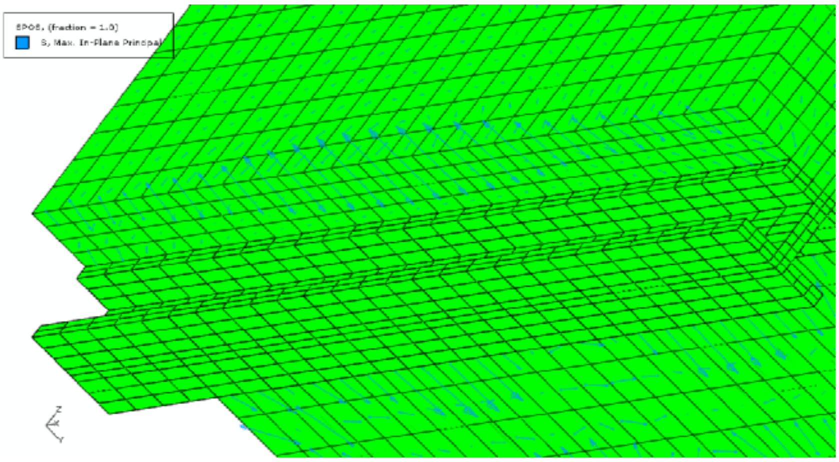


Figure 13: Maximum principal stress near junction of the container bottom membrane and container walls

5. Conclusion

The new approach in simulating a container impact with the water surface is developed. The applied methodology involves a two-way coupling between the FEA code Abaqus/Explicit and the CFD finite-volume code FlowVision, based on Direct Coupling Interface between the both codes. We show that the water compressibility has an important effect on the calculated results. This effect is limiting the pressure jump in water during the impact and makes simulation more physically consistent with real phenomena, when a container is stroking the water surface.

In addition, we show that any type of structure based on beams and ribbed stiffeners in the bottom part of the container has a big influence on a fluid flow and loads affecting the container structure during the impact.

We have to note that in the performed simulations the air between container bottom area and the water surface is not taken into account. At small angles between container bottom and the still water surface, the present air volume can influence the impact phenomena, as the air pocket can be formed in such impact region. We are planning to investigate such air presence effect in our future research in the FSI (water-air-solid) modeling.

6. References

1. Faltinsen O.M., Hydroelastic slamming, J Mar Sci Technol (2000) 5:49–65
2. Korobkin A.A., Impact of fluids and solids, Novosibirsk, (1997), p 200 (in Russian).
3. Aksenov, K. Iliine, T. Luniewski, T. McArthy, F. Popielas, R. Ramkumar "Oil Leakage Through a Valve Stem Seal", Proc. Abaqus User Conference, 2006, Boston, USA, 2006
4. A. Aksenov, A. Dyadkin, T. Luniewski, V. Pokhilko "Fluid Structure Interaction analysis using Abaqus and FlowVision", Proc. Abaqus User Conference, 2004, Boston, USA, 2004
5. Aksenov A, Dyadkin A, Pokhilko V. "Overcoming of Barrier between CAD and CFD by Modified Finite Volume Method", Proc. 1998 ASME Pressure Vessels and Piping Division Conference, San Diego, ASME PVP-Vol. 377-1., 1998
6. Aksenov A., Dyadkin A., Gudzovsky A., 1996, "Numerical Simulation of Car Tire Aquaplaning". Computational Fluid Dynamics '96, J.-A. Desideri, C.Hirsch, P.Le Tallec, M.Pandolfi, J.Periaux edts, John Wiley&Sons, pp. 815-820.
7. Belotserkovsky, 1994 "Numerical Methods in Continuum Mechanics", Moscow, Fizmatlit, 2nd edition, p.441.
8. ABAQUS Version 6.7 Documentation