"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 sub grid geometry resolution. The novel approach is that there are no auxiliary 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 test case 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.

Download the full conference paper