Computers & Fluids 88 (2013) 719–729

Contents lists available at SciVerse ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

# Parallel adaptive fluid–structure interaction simulation of explosions impacting on building structures

R. Deiterding [a,*], S. Wood [b]

[a] Oak Ridge National Laboratory, P.O. Box 2008, MS-6367, Oak Ridge, TN 37831, USA
[b] University of Tennessee – Knoxville, The Bredesen Center, 821 Volunteer Blvd., Knoxville, TN 37996, USA

ARTICLE INFO

*Article history:*
Received 14 February 2013
Received in revised form 12 May 2013
Accepted 15 May 2013
Available online 28 May 2013

*Keywords:*
Fluid–structure interaction
Adaptive mesh refinement
Cartesian embedded boundary method
Building structures
Parallelization

ABSTRACT

We pursue a level set approach to couple an Eulerian shock-capturing fluid solver with space–time refinement to an explicit solid dynamics solver for large deformations and fracture. The coupling algorithms considering recursively finer fluid time steps as well as overlapping solver updates are discussed. Our ideas are implemented in the AMROC adaptive fluid solver framework and are used for effective fluid–structure coupling to the general purpose solid dynamics code DYNA3D. Beside simulations verifying the coupled fluid–structure solver and assessing its parallel scalability, the detailed structural analysis of a reinforced concrete column under blast loading and the simulation of a prototypical blast explosion in a realistic multistory building are presented.



## 1. Introduction

The construction of efficient and scalable algorithms for simulating fluid–structure interaction (FSI) problems is an area of active research. This is particularly true for shock-driven problems e.g. [3], for which the discretizations both in fluid and solid are usually time-explicit and therefore computationally comparably inexpensive. On the other hand, major geometric complexities, such as large structural deformations [34], fracture and even fragmentation, might have to be considered. An approach to this problem is to employ an immersed or embedded boundary method in the fluid solver [24], in which moving solid structures slide through a fixed Eulerian fluid background mesh. In most cases, structured Cartesian schemes are used for embedded boundary techniques [35].

Here, we employ our verified generic Cartesian fluid solver framework AMROC [9–11,36] that implements a ghost fluid approach [15] and relies on a scalar level set function, storing the distance to the nearest boundary facet of the solid's triangulation, to represent the embedded geometry on the fluid grid [2]. To mitigate boundary approximation inaccuracies, the fluid mesh in the vicinity of the immersed boundary is refined on the fly. For coupling, a temporal splitting technique, in which solvers exchange data only at the interface between disjoint computational domains after consecutive time steps, is adopted [21,6]. Distributed memory parallelization both of the fluid and the solid mechanics solver is fully supported permitting large-scale computations of technical relevance.

The solver suite integrating AMROC with several solid mechanics solvers is named *Virtual Test Facility* (VTF) and was first released as public domain software in fall 2007 [13]. Successful FSI applications of the VTF software include, for instance, blast waves impinging on deforming viscoplastic materials modeled with a volumetric finite element method [14,26], detonation waves in combustible gases causing the fracture of piping using a thin-shell finite element approach [5], strong pressure waves in water inducing the rupture of metallic plates [12], the response of tubes made of fiber composites [29], or simulation of blunt bodies and parachutes in supersonic flows [20,18].

In this paper, we give a brief overview of the computational methodologies used and – for the first time – present FSI simulations in which AMROC is coupled through VTF modules to the serial version of the general purpose explicit solid mechanics solver DYNA3D by Hallquist and Lin [17] and used to simulate prototypical blast explosions impacting on realistic building structures. The presentation is organized as follows: In Sections 2 and 3 we give an overview of the respective aspects of the AMROC and the DYNA3D solver that are relevant to this paper. Section 4 contains a presentation of the FSI coupling methodology considering adaptive space–time refinement in the fluid solver and parallel computation of fluid and solid update steps. In Section 5, four configurations of increasing complexity are discussed: a simple elastic panel under

* Corresponding author. Tel.: +1 865 241 0782; fax: +1 865 241 4811.
*E-mail addresses:* deiterdingr@ornl.gov (R. Deiterding), swood@utk.edu (S. Wood).

0045-7930/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2013.05.009

planar shock impact, a parallel scalability assessment of a blast explosion under a realistic highway bridge, analysis of a blast corresponding to the Oklahoma bombing report [25] on a concrete column reinforced with steel, and finally, the simulation of a blast explosion event in the lobby of a complex seven-story building. The conclusions follow in Section 6.

## 2. AMROC adaptive fluid dynamics solver

### 2.1. Governing equations

The equations solved in AMROC for the purpose of this paper are the Euler equations

$$\begin{aligned} &\partial_t \rho + \nabla \cdot (\rho \mathbf{u}) = 0, \\ &\partial_t (\rho \mathbf{u}) + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) + \nabla p = 0, \\ &\partial_t (\rho E) + \nabla \cdot ((\rho E + p)\mathbf{u}) = 0. \end{aligned} \tag{1}$$

In the latter, $\rho$ is the fluid density, $\mathbf{u}$ the velocity vector, and $E$ the specific total energy. The hydrostatic pressure $p$ is given by the polytropic gas equation $p = (\gamma - 1)(\rho E - \frac{1}{2}\rho \mathbf{u}^T \mathbf{u})$ with $\gamma = 1.4$ denoting the constant adiabatic exponent of air.

### 2.2. Numerical method

To solve Eq. (1) numerically, we apply a time-explicit shock-capturing finite volume scheme based on Roe's approximate Riemann solver [32] specially hybridized with the Harten-Lax–vanLeer (HLL) scheme to ensure strict positivity preservation [7]. Second-order accuracy in smooth solution regions is achieved with the MUSCL-Hancock variable extrapolation technique.

The boundary geometry is mapped onto the Cartesian mesh with a scalar level set function $\varphi$ that stores the distance to the boundary surface and allows the efficient evaluation of the boundary outer normal in every mesh point as $\mathbf{n} = -\nabla\varphi/|\nabla\varphi$ [10]. Our implementation allows both the use of *signed* distance level set functions for representing volumetric elements [14] as well as *unsigned* distance level set functions to consider thin-shell elements [5]. In the signed distance case, a fluid cell is treated as an embedded ghost cell if its *midpoint* satisfies $\varphi < 0$. For thin-shell elements, which have a mesh only within the element midplane and implicitly assumed constant thickness $h$, we employ the condition $\varphi < h/2$ and additionally evaluate the hydrodynamic load on each thin element as the difference between the approximated pressure values at $\varphi = h/2$ in the positive and negative direction of each element's normal.

The vector of state in embedded ghost cells is then adjusted to model the boundary conditions of a non-Cartesian reflective wall moving with velocity $\mathbf{v}$ before applying the unaltered Cartesian finite volume discretization. The last step involves interpolation and mirroring of $\rho$, $\mathbf{u}$, $p$ across the boundary and modification of the normal velocity in the immersed boundary cells to $(2\mathbf{v}\cdot\mathbf{n} - \mathbf{u}\cdot\mathbf{n})\mathbf{n}$, cf. [11]. Here, we employ a dimension-wise linear interpolation operation that can decrease the number of interpolants directly near the boundary to ensure the monotonicity of the numerical solution [10].

Crucial for the performance of the overall method is the fast evaluation of the distance information, which is computationally equivalent to determining for every fluid cell the closest facet on the solid surface mesh. For this purpose, we employ a specially developed algorithm based on characteristic reconstruction and scan conversion by Mauch [23] that is used to compute the distance exactly only in a small band around the embedded structure. In the following this algorithm is denoted as *Closest Point Transform* (CPT).

### 2.3. Parallel adaptive mesh refinement

As it is characteristic for immersed Cartesian techniques, the boundary treatment described in the previous section results in some geometric approximation inaccuracies [24]. We mitigate this problem by refining the embedded boundary dynamically during the computation, typically up to the highest available resolution. A refinement criterion based on $\varphi = 0$ has been implemented for this purpose.

For local dynamic mesh adaptation we have adopted the block-structured adaptive mesh refinement (SAMR) method after Berger and Colella [4] that is tailored especially for hyperbolic conservation laws on logically rectangular grids. In this approach, finite volume cells are clustered with a special algorithm into non-overlapping computationally effective rectangular grids. Refinement levels are integrated recursively using hierarchical time step refinement. Spatial and temporal mesh widths on level $l$ are $r_l$-times finer than on level $l-1$, and a time-explicit finite volume scheme will (in principle) remain stable on all levels of the hierarchy. Here, we assume $r_l \geqslant 2$ for $l > 0$ and $r_0 = 1$.

Parallelization of the SAMR method is relatively straightforward as already in the serial algorithm subgrids are computationally decoupled by utilizing layers of halo cells. The halos on level $l$ are set either to implement physical boundary conditions, for $l > 0$ by time–space interpolation from data on level $l-1$, or by copying the data value from an overlying subgrid on $l$ (synchronization). In AMROC, we follow a rigorous domain decomposition approach and partition the SAMR hierarchy from the root level on. A careful analysis of the SAMR algorithm uncovers that the only parallel operations under this paradigm are halo cell synchronization, redistribution of the data hierarchy and the application of flux correction terms along internal refinement boundaries that impose the sum of abutting fine cell numerical fluxes on coarse grid cells [8]. Partitions with similar workload are found at runtime as the hierarchy evolves by a domain decomposition algorithm based on a generalization of Hilbert's space-filling curve [28]. The space-filling curve defines an ordered sequence on the cells of the root level that can easily be split in load-balanced portions. As such curves are constructed recursively, they are locality-preserving and therefore avoid an excessive data redistribution overhead. Further on, the surface area is small, which reduces synchronization costs. Benchmark results (not shown here) exhibit good scalability for typical SAMR fluid-only benchmarks on several thousand processors.

Finally, it is worth mentioning that the described computational techniques are equally applicable to viscous high-speed flows. For instance, Ziegler et al. [36] utilize AMROC to simulate the chemically reactive Navier–Stokes equations. In addition to an upwind scheme for convection, time-explicit conservative central difference stencils are used to approximate the viscous fluxes. The previously described embedded boundary method can easily be adjusted to impose no-slip boundary conditions but note that the resolution along the embedded boundary typically needs to be significantly finer than for Euler equations, cf. [24]. The adaptive mesh refinement methodology of AMROC provides an effective means for coping with these increased resolution requirements in space as well as in time.

## 3. DYNA3D solid mechanics solver

### 3.1. Governing equations

DYNA3D is an explicit, nonlinear, finite element code for problems where high rate dynamics or stress wave propagation effects are important. It uses a lumped mass formulation for efficiency.