International Journal of Impact Engineering 80 (2015) 65–75

Contents lists available at ScienceDirect

International Journal of Impact Engineering

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

A fluid structure interactions partitioned approach for simulations of explosive impacts on deformable structures

Vinh-Tan Nguyen [a,*], Bernhard Gatzhammer [b]

[a] Institute of High Performance Computing, 1 Fusionopolis Way, #16-16 Connexis, Singapore 138632, Singapore
[b] Technische Universitat Munchen, Institut fur Informatik, Boltzmannstr. 3, 85748 Garching, Germany

ARTICLE INFO

Article history:
Received 4 November 2014
Received in revised form
17 January 2015
Accepted 19 January 2015
Available online 14 February 2015

Keywords:
Fluid structure interactions
Shock capturing
Mesh adaptation
Unstructured grids
Air blast
Blast impact

ABSTRACT

Fluid structure interactions (FSI) has played an important role in many engineering applications ranging from aeroelasticity in aerospace engineering, wind turbines, civil engineering, bioengineering, etc. This paper focuses on FSI simulations of high speed compressible flows and its applications in studying blast impacts on flexible structures. Under the strong and high speed shock waves generated from blasts, engineering structures are deformed and possibly destructed causing damages to its surroundings. In this work, high speed compressible flows are simulated using a second order finite volume approach with HLLC flux scheme for strong shock capturing. High pressure loading from blast wave deforms structures and potentially causes severe damages. The structure responses are computed using dynamic finite element analysis. The interaction between flows and structures are coupled via exchange of information on loading conditions and structural deformations by a partitioned coupling scheme. The proposed FSI framework is applied for simulations of flat plate under explosive charge and explosion in a confined flexible cylinder. The results are well compared with literature and experimental data.

1. Introduction

Fluid structure interaction is a multiphysical phenomena in which structure responses are induced by fluid impact such as pressure or fluid forces; at the same time, dynamic responses of the structures affect fluid flow patterns and characteristics. This tightly coupled between fluids and structures plays an important role in a wide range of engineering applications in aerospace engineering, wind turbines, civil engineering, bioengineering, defense, etc. In particular, FSI is a key phenomena in studying impacts of explosions on structural components in which high speed compressible flows cause very strong impact on the surrounding structures. Understanding the impacts and mechanisms of FSI phenomena of high pressure waves generated from explosive charges on civil structures helps to better design protective structures as well as mitigation plans.

Numerical simulations increasingly serve as an effective tool for investigations of many physical phenomena. In recent years, there has been a great interest in development of numerical tools for simulations of fluid structure interactions. For FSI simulations, it requires to solve for both fluid and structure unknowns while satisfying equilibrium constraints at coupled interfaces. Fluid and structure systems are normally coupled via exchanging information at the interface. Boundary conditions at the interface assure the equilibrium in stress and displacement between fluid and structure. The continuity condition requires that the fluid mesh velocity at the interface must equal the velocity obtained from the structure response at the interface itself. Furthermore, it requires that the stress load from fluid on the interface must be transferred to the structure as external loading at grid nodes on the interface to compute the response of the structure.

The process of exchanging data between fluid and structure solvers involves many coupling strategies developed over the years. Depending on the transfer of information between fluid and structure solvers, basically there are three classes for a flow and structure coupled simulation, namely the monolithic approach, the fully coupled approach and the loosely coupled approach. In the monolithic approach, fluid properties and structure response are simultaneously advanced in time thus one has to solve for a large system of equations involved all the fluid and structure variables. Since the fluid and structure motion are viewed as a single system and synchronized while advancing in time, it enhances robustness, consistency and stability.

* Corresponding author. Tel.: +65 6419 1591; fax: +65 6467 4350.
E-mail address: nguyenvt@ihpc.a-star.edu.sg (V.-T. Nguyen).

http://dx.doi.org/10.1016/j.ijimpeng.2015.01.008
0734-743X/

Unlike the monolithic approach where the fluid and structure system are simultaneously advanced in time, in the fully coupled approach the fluid and structure system are sub-iterated until convergence. The information from fluid and structure system are transferred back and forth until the whole system is fully converged. Similar to the fully coupled approach, the fluid and structure system are also sub-iterated to convergence in each time step in the loosely couple approach but they only exchange data a few times in each step. It is less expensive than the other approaches by saving the computational cost per each time step, however solutions of fluid and structure system are usually staggered in time and it could lead to temporal errors. Due to nature of loosely and fully coupled approach where one only needs to solve for each continuum separately, they are also referred to as partitioned approaches. One of the most practical advantages of the partitioned approach is the reusability of existing solvers.

In this work, we proposed a coupling paradigm for fluid structure interaction simulations of high speed compressible flows and deformable structures. A second order edge-based finite volume solver [9] was used for simulations of high speed compressible flows. Strong shocks are resolved by employing a robust shock capturing scheme with second order HLLC flux and solution mesh adaptation. Structural responses are computed using finite element analysis with a wide range of element types and constitutive models for different materials. The coupling of fluid and structure solver is realized by a general coupling environment [2] which maintains the full flexibility of the partitioned approach and provides a variety of coupling and data mapping functionalities as well as clearly defined interfaces for the implementation of additional variants. The proposed framework was applied for simulations of a square plate and thin cylinder under explosive load impact from air blasts. Results from the current simulations is compared with experimental and literature data to show the effectiveness and accuracy of the proposed method for this type of applications.

2. Governing equations

Consider a generalized fluid structure interaction problem in a three dimensional domain $\Omega = \Omega^f \cup \Omega^s \in \mathbb{R}^3$ shown in Fig. 1 where fluids occupies the domain Ω^f and structures occupies the domain Ω^s with appropriate boundary conditions applied at the respective domain boundaries. The interaction between fluids and structures is described at the interface $\Gamma^i = \Omega^f \cap \Omega^s$. Solutions to the FSI problem is sought by solving for fluid unknowns, structural responses subjected to coupled relations between fluids and structures at the interface.

2.1. High speed compressible flows solver

The unsteady compressible flows are governed by the time-dependent, Euler equations expressed in the conservative form as

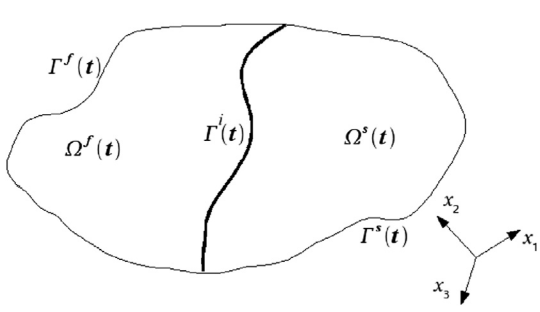

Fig. 1. A general fluid structure interaction description of fluid domain Ω^f coupled with structure domain Ω^s at the interface Γ^i.

$$\boldsymbol{w}_t + \nabla \cdot \boldsymbol{F}(\boldsymbol{w}) = \boldsymbol{0}, \tag{1}$$

on a three−dimensional domain $\Omega^f \subset \mathbb{R}^3$ with the appropriate initial and boundary conditions applied on the domain boundary $\Gamma^f \equiv \partial\Omega^f = \Gamma^f_D \cup \Gamma^f_N$.

$$\begin{aligned} \boldsymbol{w}(\boldsymbol{x},0) &= \boldsymbol{w}_0(\boldsymbol{x}) \\ \boldsymbol{w}(\boldsymbol{x},t) &= \boldsymbol{g}_D(\boldsymbol{x},t) \quad \boldsymbol{x}\in\Gamma^f_D \\ \nabla_{\boldsymbol{n}}\boldsymbol{w}(\boldsymbol{x},t) &= \boldsymbol{g}_N(\boldsymbol{x},t)\cdot\boldsymbol{n} \quad \boldsymbol{x}\in\Gamma^f_N \end{aligned}. \tag{2}$$

The unknown vector $\boldsymbol{w}$ of the conservative variables and inviscid flux tensors $\boldsymbol{F}$ are given by

$$\boldsymbol{w} = \begin{pmatrix} \rho \\ \rho v_1 \\ \rho v_2 \\ \rho v_3 \\ \rho\varepsilon \end{pmatrix}, \quad \boldsymbol{F}_j = \begin{pmatrix} \rho v_j \\ \rho v_1 v_j + p\delta_{1j} \\ \rho v_2 v_j + p\delta_{2j} \\ \rho v_3 v_j + p\delta_{3j} \\ v_j(\rho\varepsilon + p) \end{pmatrix}, \quad \forall j.$$

Here ρ denotes the fluid density, v_i the i'th component of the velocity vector and ε the specific total energy. The system is closed by assuming the gas to be calorically perfect, thus setting $p = \rho RT$ and the ratio of the specific heats, $\gamma = c_p/c_v$ is set to $\gamma = 1.4$ for air at standard conditions.

The computational domain $\Omega^f(t)$ is subdivided into a set of non-overlapping tetrahedral elements $\mathscr{T}^f_h$ using a Delaunay mesh generation process with automatic point creation [13]. In order to account for displacement of coupled interfaces due to fluid-structure interaction the arbitrary Eulerian-Lagrangian (ALE) approach is employed to allow control volumes to move independently of the flows. The governing equation can then be written in ALE integral form as follows,

$$\int_{\Omega} \frac{\partial \boldsymbol{w}}{\partial t} \mathrm{d}\boldsymbol{x} + \int_{\partial\Omega} (\boldsymbol{F}_j - \boldsymbol{v}_j \boldsymbol{w})\boldsymbol{n}_j \mathrm{d}\boldsymbol{x} = 0, \tag{3}$$

where the conventional summation is employed, $\boldsymbol{v}$ is the mesh velocity and $\boldsymbol{n}$ is the outward unit normal vector to $\partial\Omega$. This governing equations of the flows are discretised using second order edge−based, vertex center finite volume method [9].

The contribution of the inviscid flux over the control volume surface for node I is then computed as

$$\int_{\partial\Omega_I} \boldsymbol{F}_j n_j \mathrm{d}\boldsymbol{x} \approx \sum_{J\in\Lambda_I} C^{IJ} \mathscr{F}^{IJ} + \sum_{J\in\Gamma^B_I} D^{IJ} \mathscr{F}^{I}, \tag{4}$$

where Λ_I denotes the set of nodes connected to node I by an edge and n_j is the outward unit normal vector to $\partial\Omega$. The inviscid numerical flux F^{IJ} in IJ direction is obtained from solving local Riemann problems at the facets of node I's control volume, $\mathscr{F}^{IJ} \approx \boldsymbol{F}\cdot\boldsymbol{n}^{IJ}$. In the above formulation, C^{IJ}_j and D^{IJ}_j are the value of edge coefficients for internal and boundary edges, respectively.

In this work, the intercell flux F^{IJ} is computed using compact Harten-Lax-van Leer (HLLC) flux scheme [9]. The HLLC scheme is complemented by second order reconstruction of Riemann states thus obtaining second order accuracy. As the flows develop strong shock waves, various slope limiters were implemented to stabilize solutions and overcome instabilities due to the high order approximation.

The discrete ALE term for node I at time $t = t^n$ can be computed as