

Application of CEL Method for Simulation of Multiphysics Events in Automobiles

2011-01-0793

Published
04/12/2011Ranjit Tanaji Babar, Varma Pakalapati and Vidyadhar Katkar
Tata Technologies Ltd

Copyright © 2011 SAE International

doi:10.4271/2011-01-0793

ABSTRACT

In automobiles, there are various multiphysics (specifically fluid structure interaction) events taking place which are very important from vehicle operations. Examples are - oil splashing in engine, water spraying on the windscreen, fuel sloshing in a tank etc. The simulation of such events becomes important in the design stage in order to study their proper functioning before the prototypes are made.

This paper enlightens the systematic procedures developed for the simulation of such events using coupled Euler-Lagrangian method available in commercial finite element explicit codes.

These simulations are very time consuming because of very small time steps and very large cycle time. To overcome this problem an attempt is made to use rigid bodies and a low bulk modulus fluid to speed up the simulation exponentially. These quick simulations can be used for early design iterations and final designs can be revalidated with flexible bodies and correct bulk modulus.

Based on this simulation method, following case studies are presented.

- Oil splashing in an engine
- Fuel sloshing in fuel tank
- Fluid Motion Study
- Low fuel level management in a vehicle

INTRODUCTION

The study of Fluid-structure interaction (FSI) events in an automobile is an interesting topic but difficult from simulation point of view. In past, such problems were solved

sequentially i.e. the fluid domain was solved differently and after the completion of fluid simulation, its response on the structure was evaluated.

Recently such problems are solved using coupled simulation capability (referred to as co-simulation) available in various commercial softwares. There are various methods available for simulation of such problems namely, coupled Eulerian - Lagrangian (CEL) approach; Arbitrary Lagrangian - Eulerian (ALE) approach, Smooth Particle Hydrodynamics approach (SPH) and co-simulation of general purpose CFD code and FEA code. [1]

Each method has its advantages and disadvantages. Engineers have tried solving multiphysics problem using various methods and have presented differences between them [1, 5, 6, 8]. It is found from the literature that suitability of a particular method varies from application to application.

In this paper, CEL method is used for the simulation of FSI problems in automotive domain. It is observed that, this method is well suited for the presented applications. The authors have used Abaqus software for simulation of these events.

BASICS OF LAGRANGIAN AND EULERIAN METHOD

LAGRANGIAN METHOD

In the Lagrangian method the spatial part of the domain is discretized by 1-D, 2-D, 3-D or discrete elements. Lagrangian elements are constant mass elements and a finite element mesh is attached to the material and these elements deform as the material starts to deform. These finite elements are connected by the common grid points. The material mass and velocity is defined at the grid points. Forces such as inertia,

Table 1. comparison of Lagrangian and Eulerian methods

Feature	Lagrangian Methods	Eulerian Methods
GRID	Attached to the moving material	Fixed in space
TRACK	Movement of any point on material	Mass, momentum and energy flux across grid nodes and mesh cell boundary
TIME HISTORY	Easy to obtain time-history data at a point attached on materials	Difficult to obtain time history-data at a point attached on materials
MOVING BOUNDARY AND INTERFACE	Easy to track	Difficult to track
IRREGULAR GEOMETRY	Easy to model	Difficult to model with good accuracy
LARGE DEFORMATION	Difficult to handle	Easy to handle

stiffness, interaction and external forces act on the grid points. Stresses are defined at the integration points or the element centroid.

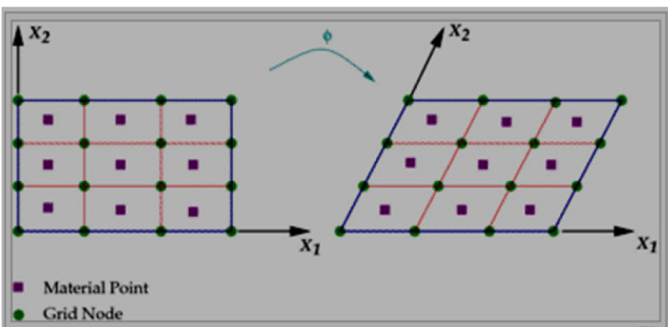


Fig 1. Lagrangian Method

In this method as the boundary nodes remain on the boundary itself; boundary conditions and interface conditions can be easily defined. Since the mesh deforms with the material, severe mesh deformations can occur deteriorating mesh quality. Due to all these peculiarities, this method is suited for problems in which the mesh deformations are less, particularly in case of metal structures [9].

EULERIAN METHOD

In the Eulerian method the spatial part of the domain is discretized by volume elements. In this method only brick elements (8-noded hexahedral elements) are available. Eulerian mesh is fixed in time and space. Eulerian elements are constant volume elements and the grids have no degrees of freedom. In the Eulerian method, the material moves from element to element and allows severe deformations of the mesh since the material can freely flow inside the Eulerian mesh. The material state at each point of the Eulerian domain is defined by velocity, density, specific internal energy and

stress tensor at any point of time. These variables relate to each other by conservation of mass, momentum, energy equations and equation of state. The solution in this method is computed in space using control volume method.

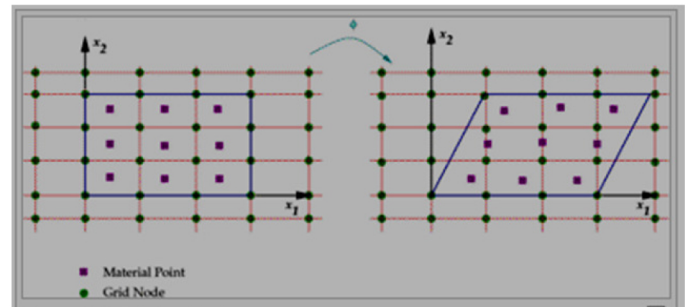


Fig 2. Eulerian Method

Mesh boundary nodes and material boundary may not coincide in this method hence boundary conditions on the Eulerian elements are difficult to apply. There are no mesh distortions in this method as the mesh is fixed in space. However the domain that needs to be modeled is larger since the material should not leave the body. Because of all these peculiarities, this method is best suitable in the problems where the severe material deformations are possible particularly in case of fluids [9].

COMPARISON OF EULERIAN AND LAGRANGIAN METHODS

The following table summarizes the difference between these two methods with respect to different features.

COUPLED EULERIAN-LAGRANGIAN (CEL) METHOD FOR FLUID STRUCTURE INTERACTION

Fluid-structure interaction events can be effectively modeled using CEL formulation available in various commercial FE codes. In CEL formulation, Lagrangian domain deals with the deformations of the structure part and Eulerian domain deals with the fluid part of the problem. These two domains interact with each other with contact definition between two. General contact algorithms in explicit codes, enforce contact between Eulerian materials and Lagrangian surfaces. All or individual Eulerian surfaces can be specified in the contact domain with Lagrangian surfaces. Contact interactions between Eulerian materials and interactions due to Eulerian material self-contact, are handled by the Eulerian formulations. [2]

As explained earlier; the fundamental equations governing the motion of rigid and deformable bodies are those of motion, continuity and energy. Finite element explicit codes solve these equations in an explicit dynamics analysis procedure. These equations are listed below:

CONTINUITY EQUATION (CONSERVATION OF MASS)

$$\frac{D\rho}{Dt} + \rho \nabla \cdot \mathbf{v} = \frac{\partial \rho}{\partial t} + \mathbf{v} \cdot \nabla \rho + \rho \nabla \cdot \mathbf{v} = 0 \quad (1)$$

Where, $\frac{D}{Dt}$ denotes the total derivative and $\frac{\partial}{\partial t}$ denotes the partial derivative. 'del' (∇) is the gradient / differential operator and 'del dot' ($\nabla \cdot$) is the divergence operator.

EQUATION OF MOTION

$$\begin{aligned} \rho \frac{D\mathbf{v}}{Dt} &= \nabla \cdot \boldsymbol{\sigma} + \rho \mathbf{b} \\ \frac{D(\mathbf{v})}{Dt} &= \frac{\partial(\mathbf{v})}{\partial t} + \mathbf{v} \cdot \nabla(\mathbf{v}) \end{aligned} \quad (2)$$

ENERGY EQUATION

$$\rho \frac{DE}{Dt} = \boldsymbol{\sigma} \cdot \dot{\boldsymbol{\epsilon}} + \rho \dot{Q} \quad (3)$$

Where $\dot{\boldsymbol{\epsilon}} = \frac{1}{2}(\nabla \mathbf{v} + \nabla \mathbf{v}^T)$ is the strain rate (commonly referred to as rate of deformation tensor). For incompressible

flows, variations of density within the flow are negligible which results in $\rho \nabla \cdot \mathbf{v} \cong 0$

Finite element solver solves the continuity, motion and energy equations. The notion of a material (solid or fluid) is introduced when specific constitutive assumptions are made. The choice of a constitutive law for a solid or a fluid will reduce the equation of motion appropriately. The various constitutive choices for fluids are : - i) Navier- Stokes equations for compressible and incompressible fluids with and without Bulk viscosity ii) Euler equations in case of inviscid fluids, ideal gases. [4]

TIPS FOR SOLVING CEL PROBLEMS

In the present study, Simulia - Abaqus is used as a finite element solver for CEL method in explicit domain. Following tips and techniques make the CEL simulations more accurate and fast.

MATERIAL MODELING IN EULERIAN DOMAIN

As the material strains in the Eulerian domain tends to increase far beyond, the material data needs to be defined over the extended strain range to avoid simulation termination due to severe mesh distortions and mesh tangling (negative volumes). In case of fluid flow problems such as fluid sloshing and hydroplaning problems (applications in automotive domain); equation of state material models is recommended to use. Fluid viscosity should be accounted by introducing shear properties. [2]

The compressibility of the fluid shall be used close to the actual physical value by choosing the correct bulk modulus value.

CONTACT BETWEEN LAGRANGIAN AND EULERIAN DOMAINS

Eulerian mesh needs to be modeled in all areas where the possible fluid flow will occur during the course of the simulation event. Also it is recommended that the Eulerian domain has to be extended beyond the Lagrangian domain by at least 1-2 elements.

General contact formulation allows the contact between all the Eulerian and Lagrangian elements along with self contacts. In order to reduce the simulation time non-essential Lagrangian surfaces can be excluded from the general contact domain. In case of fluid problems, the refinement of Eulerian elements helps to reduce the penetrations and leakage of fluid beyond the Lagrangian domain. Typically Eulerian mesh size up to 2 times less than minimum element size of the Lagrangian meshes works well in most of the problems discussed in this paper.



Initial volume fractions of the Eulerian domain elements need to be defined for representing initial volume of fluid at the start of simulation. In case of complex geometries, this becomes a difficult task and use of appropriate preprocessors such as ABAQUS-CAE becomes necessary.

HOURLASS CONTROLS OF EULERIAN DOMAIN

As the Eulerian elements are reduced integration solid elements, hourglass control is required for solution. In most of the solvers, the default hourglass settings (viscous hourglass control in case of reduced integration Eulerian elements) work well.

For flow-type problems using the EOS models, viscous hourglass control causes the fluid to behave more like a sponge or foam. In such cases for getting realistic fluid behavior, it is better to set the displacement hourglass scaling factor to 0 instead of 1.0. In the present study it is observed that there is no significant difference in the results of Lagrangian domain elements by changing these settings but becomes useful in the cases where the fluid flow pattern is of interest. Standard checks on hourglass energy are required to be done.

TECHNIQUES TO REDUCE THE SOLUTION TIME

In case of quasi-static problems and involving strain-rate independent material properties, loads can be ramped to the actual value in artificially shorter time to reduce the total event time. Higher material density for elements can be used to increase the stable time increments. In both of these cases the kinetic energy has to be low in comparison to the internal energy.

In some models, such as low frequency tank sloshing, fluid compressibility condition can be relaxed. It is observed that results did not affect significantly. This is achieved by reducing the 'bulk modulus' which reduces the speed of sound and correspondingly increase the stable time increments. In such cases the solution must be verified carefully. It is recommended to check that the volume of fluid does not change significantly. The energy due to the change of volume of all Eulerian elements has to be much lower than internal energy in the model.

Eulerian Element Size Selection

Total simulation time depends on the stable time increment which is a direct function of element length, density and elastic modulus of material. Also the significant simulation time is consumed in resolving the Eulerian-Lagrangian contact. This time is directly dependent on the number of elements in the general contact domain. This cost can be

reduced by excluding non-essential Lagrangian surfaces from the contact domain.

Choosing correct Eulerian element size helps to reduce the solution time. It is recommended to use the Eulerian element size equal to the smallest Lagrangian element size to start with. In case the penetrations or fluid leakages occur, reduce the element size by ~20 % and check for the penetrations again. These iterations can be done in a model with rigid Lagrangian mesh which requires less simulation time as compared to flexible Lagrangian mesh. Use of rigid Lagrangian mesh is explained below in one of the case studies. After the verification of the contact penetrations, final solution with flexible Lagrangian elements can be given.

SIMULATION OF OIL SPLASHING IN AN ENGINE

There are various situations in the vehicle operation conditions when the oil in the engine is splashed. These situations include

1. Vehicle is going on a gradient and the crankshaft dips in the oil in the oil sump.
2. During the braking or the acceleration of the vehicle.
3. Balancer shaft used in the engine to reduce vibrations caused by the rotation of the crankshaft; is submerged in the oil in the sump and rotation of the balancer shaft causes splashing of oil.

In all of the above situations, as the crankshaft and balancer shaft rotates; oil is stirred in the oil pan causing bubbles in the oil, formation of foam and oil is splashed on the walls of the crankcase. This causes to reduce the lubricity of the oil. Hence excessive splashing of oil in the engine crankcase is not desired.

The simulation of these events helps to identify the probable problems during design stage. To simulate such multi-physics event, coupled Eulerian-Lagrangian method is suited where oil is modeled in Eulerian domain while all other engine aggregates are modeled in Lagrangian domain.

SIMULATION MODEL

To simulate this event entire engine assembly consisting of following components is modeled.

- Cylinder block, crankshaft assembly consisting of crankshaft, flywheel, damper pulley
- Balancer shaft assembly consisting of balancer shafts, gears and housing
- Piston, piston pin, conrod assembly consisting of connecting rod, cap and sleeves

