

# LS-DYNA<sup>®</sup> 980 : Recent Developments, Application Areas and Validation Process of the Incompressible fluid solver (ICFD) in LS-DYNA

## Part 1

Facundo Del Pin  
Iñaki Çaldichoury

*Livermore Software Technology Corporation  
7374 Las Positas Road  
Livermore, CA 94551*

### Abstract

*LS-DYNA version 980 will include CFD solvers for both compressible and incompressible flows. The solvers may run as standalone CFD solvers where only fluid dynamics effects are studied or they could be coupled to the solid mechanics and thermal solvers of LS-DYNA to take full advantage of their capabilities in order to solve fluid-structure interaction (FSI) problems.*

*This paper will focus on the Incompressible CFD solver in LS-DYNA (ICFD) and will be divided in two parts. Part one will present some advanced features of the solver as well some recent developments or improvements. Part two will provide some insight on the validation process that is currently under way in order to better understand the present capabilities and state of advancement of the solvers. Several test cases and results will be presented that will highlight several main features and potential industrial application domains of the solvers. The future steps and the challenges that remain will also be discussed.*

### 1- Introduction

LS-DYNA version 980 aims to solve complex multi-physics problems involving fluids, electromagnetism or chemistry interacting with the solid mechanics and thermal solvers of LS-DYNA. As the development of these solvers progresses, several verification, validation and benchmarking tests have been conducted both internally at LSTC and externally by beta testing users in order to track bugs and improve numerical accuracy. This paper will focus on the incompressible flow solver (ICFD) and will be divided in two parts. Part one will present some advanced features of the solver as well as some recent developments or improvements for users already familiar with the solver. Part two will provide some insight on the validation process that is currently under way in order to better understand the present capabilities and state of advancement of the solvers. The test cases presented will be divided and organized by their application domains. The idea behind this procedure is to give future users an insight of the solver's capabilities as well as a better understanding of its various potential industrial

applications. A brief description of each model will be given as well as some of the main results obtained. In the future, a more complete description of these test cases will be made available for users who would wish to try and reproduce them. It is also interesting to note that a similar procedure is being conducted for the Electromagnetism (EM) and compressible fluid (CESE) solvers.

## 2- Summary of the solver's features

The incompressible flow solver is based on state of the art Finite Element technology applied to fluid mechanics. It is fully coupled with the solid mechanics solver. This coupling permits robust FSI analysis via either an explicit technique when the FSI is weak, or using an implicit coupling when the FSI coupling is strong (See Figure 4 for an application example). In addition to being able to handle free surface flows, there is also a biphasic flow capability that involves modeling using a conservative Eulerian level set interface tracking technique (See Figure 5 for an application example). Basic turbulence models are also supported (See Figure 6 for an application example). Finally, a thermal solver for solving the heat equation in the fluid is included that is strongly coupled via a monolithic approach to the LS-DYNA thermal solver thus allowing the resolution of complex conjugate heat transfer problems (See Figure 7 for an application example). Further details on the solver's features can be found in [1].

This solver is the first in LS-DYNA to make use of an automatic new volume mesher based on the work by [2] that greatly simplifies the pre-processing stage. For FSI simulations, the solver used an Arbitrary Lagrangian Eulerian (ALE) approach for mesh movement. In the cases where FSI simulations result in large displacements the solver can automatically re-mesh to keep an acceptable mesh quality. In addition, during the time advancement of the incompressible flow, the solution is adaptively re-meshed as an automatic feature of the solver. Another important feature of the mesher is the ability to create boundary layer meshes. These anisotropic meshes become a crucial part of the model when shear stresses are to be calculated near fluid walls.

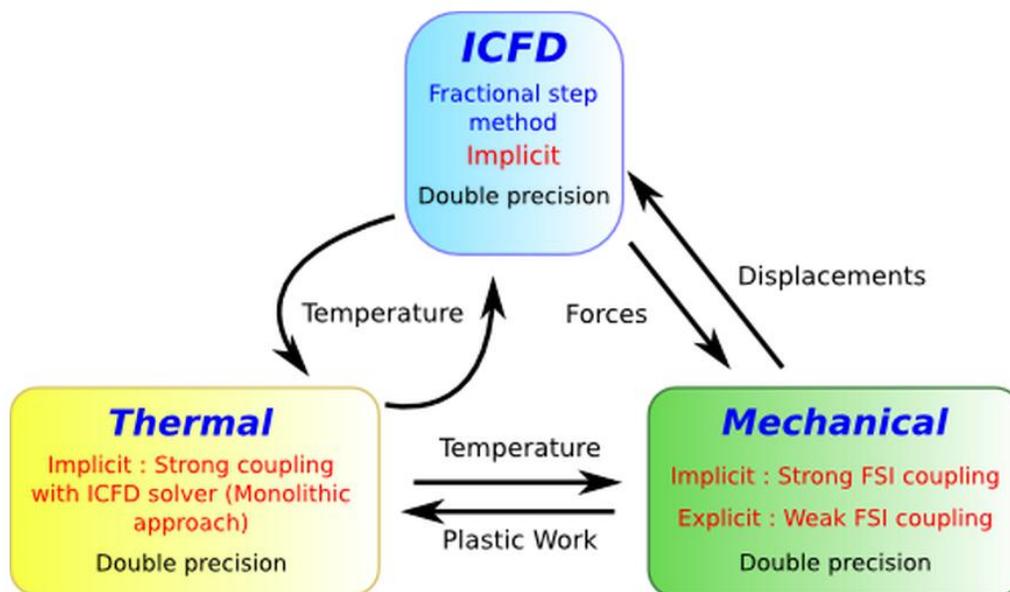


Figure 1 ICFD solver coupling with Solid Mechanics and thermal LS-DYNA solvers

### 3- Recently added features

#### 3-1 Features for the Volume Mesher

During the geometry set up, the user can define surfaces that will be used by the volume mesher to specify a local mesh size inside the volume. Two methods are available:

- In the first method, a surface with the specified element size must be defined in the input deck and will be read along with the other initial surfaces. This surface must be entirely defined in the volume mesh but does not need to be necessarily closed (water tight). The nodes of this specific surface will directly be included when building the initial volume mesh. A linear interpolation will then be used to define the element sizes between this specifically added surface and the classic boundary surface meshes defining the fluid volume enclosure.
- In the second method, a local element size can be defined by the user in specific zones corresponding to given geometrical shapes (box, sphere or cylinder). This element size will be used as reference in the specified zone when building the initial volume mesh. This zone does not necessarily need to be entirely defined in the fluid volume mesh. However, in cases where a boundary surface mesh crosses or is entirely defined in the zone where the mesh size is imposed, it is advised to use a similar element size for the shape zone and the boundary surface mesh in order to keep a good quality mesh.

The user can also trigger an automatic remeshing where the solver will use an a-posteriori error estimator to compute a new mesh size bounded by the user to satisfy a maximum percentual error. The error estimator and adaptive remeshing procedure is based on the work by [3]. This adaptive remeshing is now fully compatible with the “boundary layer mesh” feature that consists in specifying several anisotropic elements to be added to the boundary layer in order to better represent close-to-the-wall effects.

#### 3-2 The Level Set Method for free surface problems

A large collection of fluid problems involves moving interfaces. Applications include air-water dynamics, breaking surface waves and solid bodies penetrating in fluids. In many such applications, the interplay between the interface dynamics and the surrounding fluid motion is subtle, with factors such as density ratios and temperature jumps across the interface, surface-tension effects, topological connectivity, and boundary conditions playing significant roles in the dynamics. The solver uses a level set method based on [4], a fast and reliable technique in order to track and correctly represent moving interfaces. When defining an interface between fluid and vacuum or between two fluids, a classic way to proceed would be to assign some nodes on the interface and to move them by their fluid velocity values  $V$  through the grid. This Lagrangian formulation would not be too hard to accomplish [4] if the connectivity does not change and the surface elements are not distorted too much. However, even small velocity fields could cause large distortion of the interface elements and the accuracy of the method can deteriorate quickly if a frequent and regular re-meshing of the domain is not applied. This need for frequent re-meshing implies higher computer costs and less scalability when running with multiple processors. In order to avoid these problems, an implicit distance function called the level set function  $\varphi$  will be used to define the interface ( $\varphi = 0$  at the interface with  $\varphi$  changing signs on

the two sides of the boundary, its absolute value increasing with the distance to the interface). In order to represent the evolution of the interface, a simple convection equation will be applied to  $\phi$ . It is an Eulerian formulation of the interface evolution [4] since the interface is captured by the implicit function  $\phi$  as opposed to being tracked by the displacement of the interface nodes.

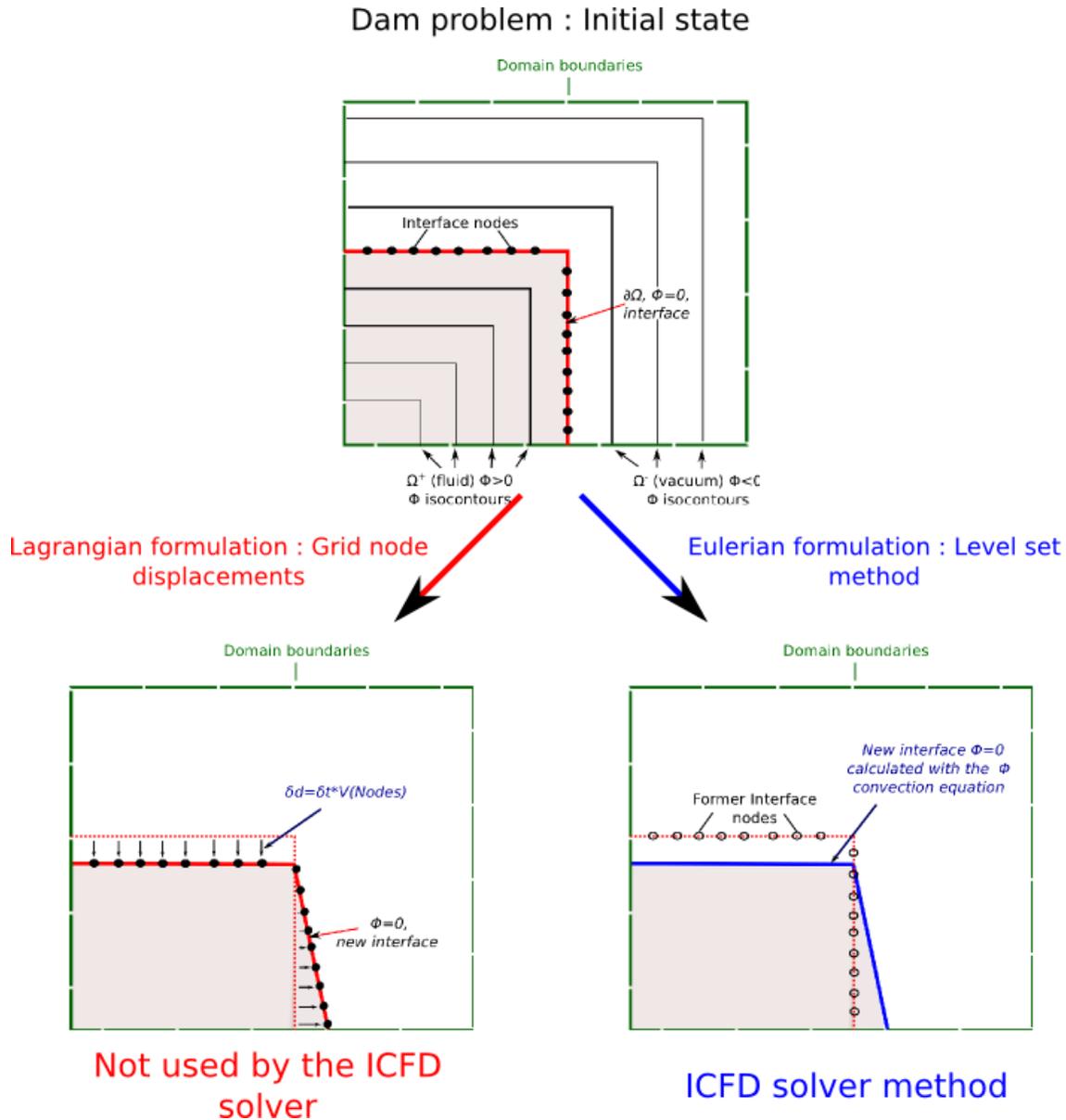


Figure 2 Free surface dam breaking problem: description of the different interface handling methods

#### 4- MPP scalability of the ICFD solver

MPP (short for Massively Parallel Processing) is a type of computing available for LS-DYNA that uses many separate CPUs running in parallel each with their own memory rather than a shared memory to execute a single analysis. In order to solve large implicit CFD analyzes, it is important to provide a good CPU scalability in order to accelerate the analysis and save some computational time. A numerical model of a flow around a bluff body of around 2 million

degree of freedom was run as a stand-alone CFD model. An FSI analysis has also been performed on the same model bringing the total number of degrees of freedom to around 3.5 million. Keeping in mind that the ICFD solver runs in implicit, the results show good speedup capabilities: of 40 for 128 cpus in the CFD only case and a speedup of 55 for 128 cpus in the case of FSI. For the next development cycle further improvements will be implemented.

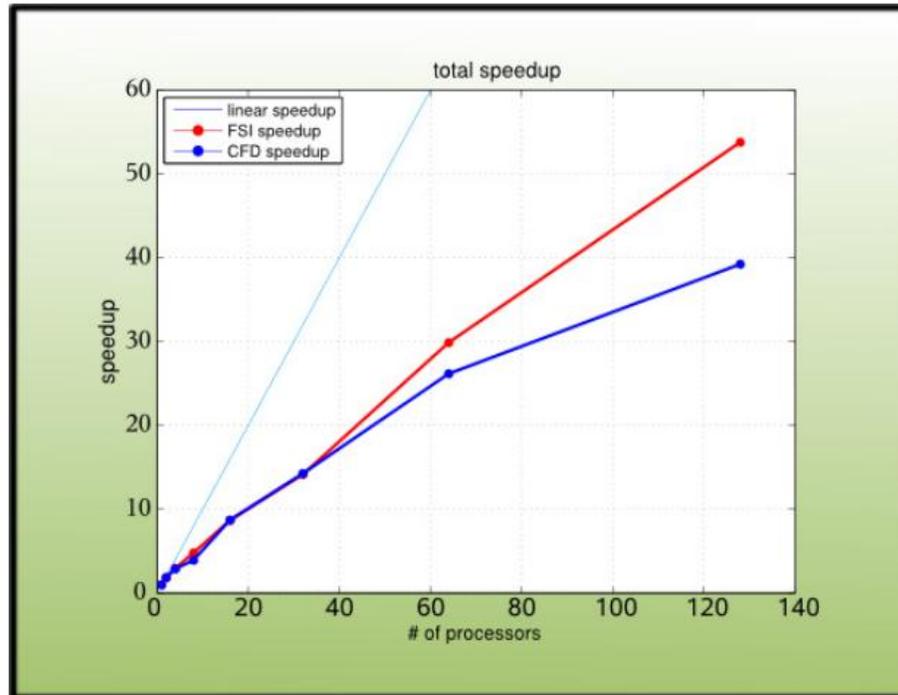


Figure 3 MPP scalability results for CFD only (approx. 2 million element model) and FSI (approx. 3.5 million element model) analysis.

## 5- Conclusion-Part 1

In the first part of this paper, a summary of the various solvers' features was given as well as a description of recent developments for users already familiar with the solver. Results obtained for MPP scalability was also given in order to give users a first feeling of what speed ups to expect. The second part of the paper will present some of the main application domains of the solver and give some information and results on validation test cases.

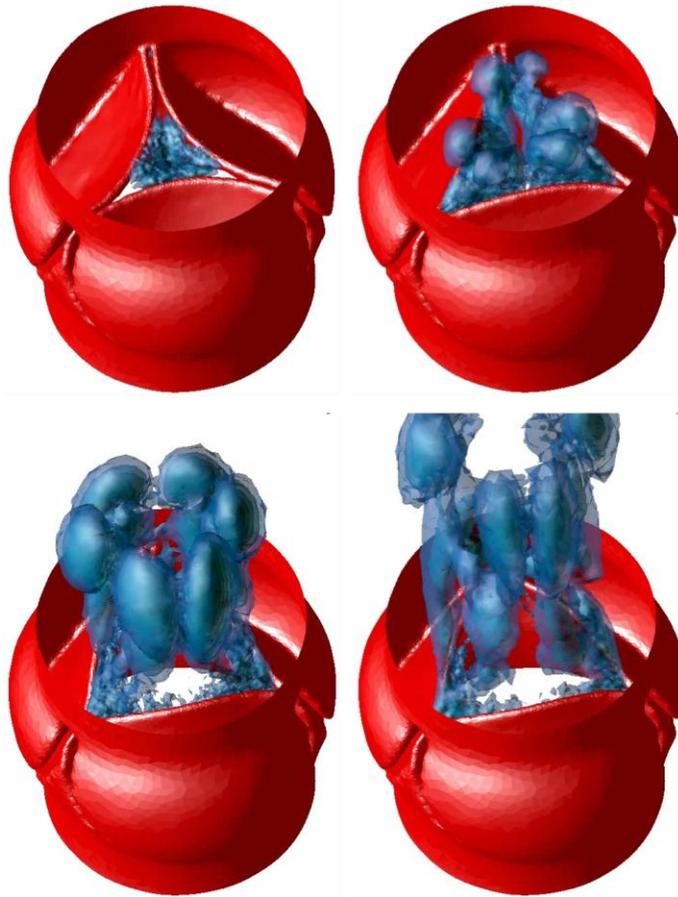


Figure 4 FSI problem: Heart valve opening due to the fluid pressure.

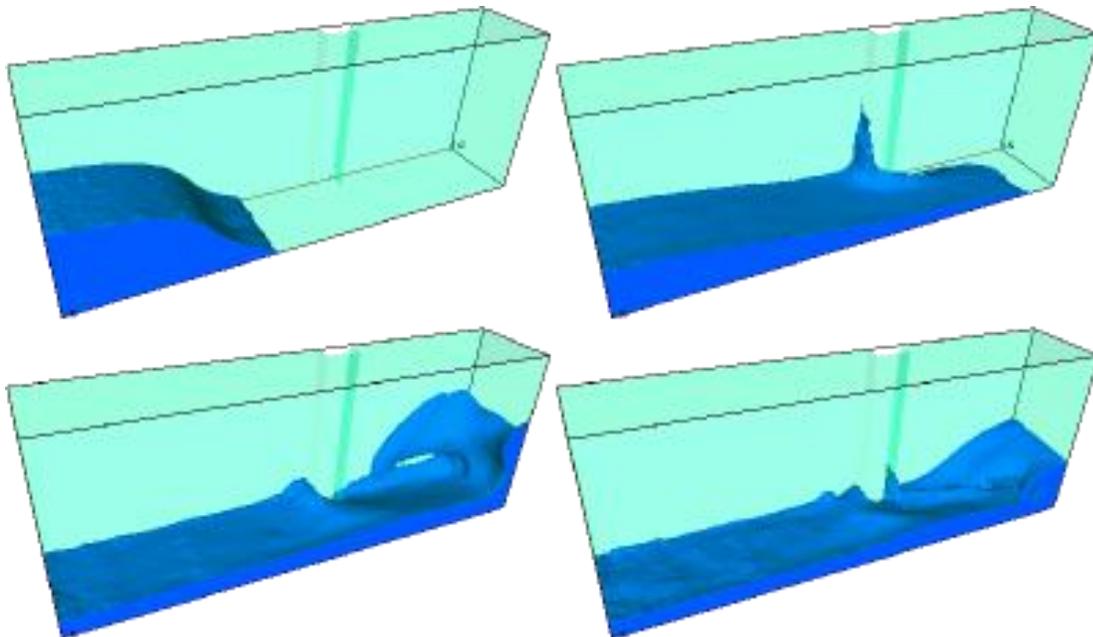


Figure 5 Free surface problem: Wave impact against a pillar structure

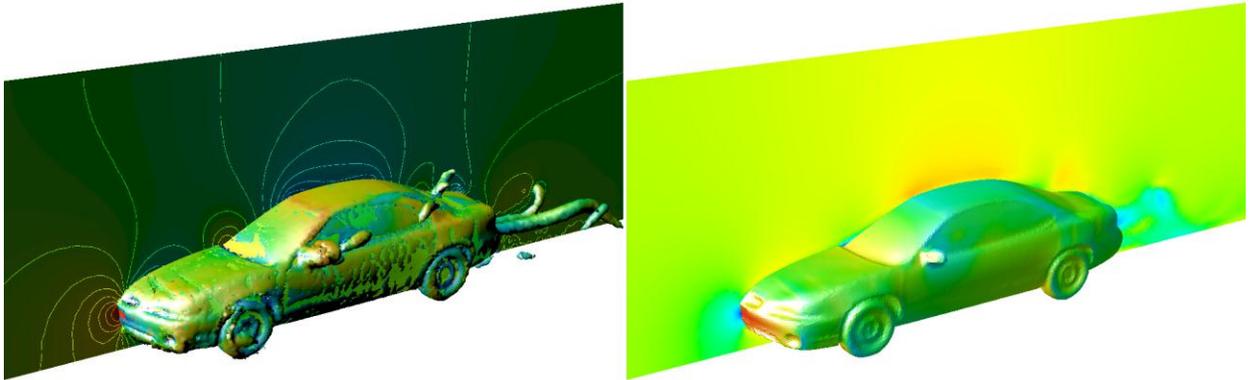


Figure 6 Turbulent CFD problem: Drag calculation around a vehicle.

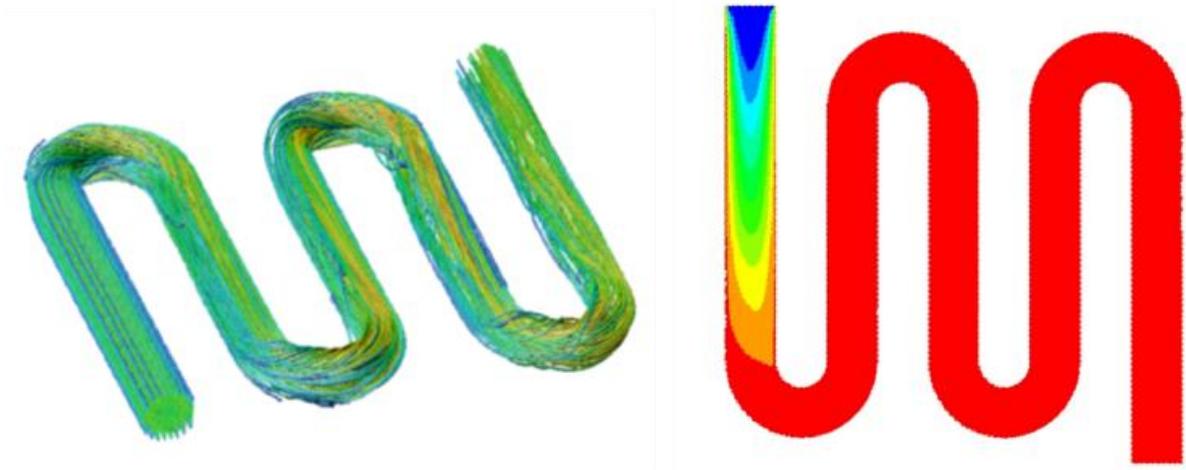


Figure 7 Thermal problem: 3D flow in a pipe. Velocity streamlines and temperature fringes.

### References

- [1] I. Çaldichoury and F. Del Pin, "ICFD theory manual," LSTC, 2012.
- [2] P. J. Frey and P.-L. George, Mesh Generation, Oxford-Paris: Hermes Science, 2000.
- [3] O. Zienkiewicz and J. Zhu, "A simple error estimator and adaptive procedure for practical engineering analysis," *International Journal for Numerical Methods in Engineering*, vol. 24, pp. 337-357, 1987.
- [4] S. Osher and R. Fedkiw, Level Set Methods and Dynamic Implicit Surfaces, Springer, 2003.
- [5] S. Idelsohn, F. Del Pin, R. Rossi and E. Oñate, "Avoiding the instabilities caused by added mass effects in fluid structure interaction problems for pressure segregation and staggered approaches," *Int. Journal. Num. Meth. Eng.*, 2009.