Numerical Simulation of a Non-Linear Coupled Fluid-Structure Problem With Implicit and Explicit Coupling Procedures

Jean-François Sigrist, Driss Abouri

► To cite this version:

Jean-François Sigrist, Driss Abouri. Numerical Simulation of a Non-Linear Coupled Fluid-Structure Problem With Implicit and Explicit Coupling Procedures. ASME 2006 Pressure Vessels and Piping/ICPVT-11 Conference, Jul 2006, Vancouver, France. 10.1115/PVP2006-ICPVT-11-93107 . hal-01911095

HAL Id: hal-01911095

<https://hal.science/hal-01911095>

Submitted on 2 Nov 2018

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

NUMERICAL SIMULATION OF A NON-LINEAR COUPLED FLUID-STRUCTURE PROBLEM WITH IMPLICIT AND EXPLICIT COUPLING PROCEDURES

Jean François SIGRIST
Service Scientifique et Technique
DCN Propulsion
44620 LA MONTAGNE, France
jean-francois.sigris@dcn.fr

Driss ABOURI
CD-adapco
31, rue Delizy
93698 PANTIN Cedex, France
driss.abouri@fr.cd-dapco.com

ABSTRACT

The present paper deals with the numerical simulation of a non-linear coupled fluid-structure problem with finite element/finite volume coupled technique and mainly focus on the time coupling algorithm: the coupling procedure lies on a staggered explicit or implicit strategy.

A simple coupled case is studied; namely a non-linear elastic beam coupled with an incompressible fluid with free surface effects. The structure problem is solved with a finite element approach. The non-linear discrete structure problem is integrated in time with the non-linear Newmark scheme with a fixed-point procedure. The fluid problem is solved with a finite volume approach. The non-linear discrete fluid problem is solved with the PISO (Pressure Implicit by Splitting Operator) algorithm. The coupled problem is solved with two different approaches.

At first, an explicit approach is proposed, based on a step by step alternate integration of the structure and fluid non-linear problems. As expected the method is found to be numerically dissipative and potentially unstable. An implicit approach is then proposed, using inner coupling iterations. The proposed algorithm is proved to be stable and less dissipative.

Implicit and explicit approaches are presented. Numerical calculations are performed on the elementary studied case. Results are presented and compared for the two time-integration procedures, highlighting the numerical qualities of each procedure in terms of numerical damping, stability and precision.

1. INTRODUCTION

The present paper deals with the numerical simulation of coupled fluid-structure problem, in which the coupling process is based on mechanical exchanges between each sub-problem. In the present study other fluid-structure coupling phenomena, such as thermal effect, are not taken into account. At the fluid

structure interface, fluid and structure are coupled via fluid forces and structure displacement, as sketched by Fig. 1.

Over the past years many numerical methods have been developed in order to solve linear and non-linear coupled fluid-structure problems [13] [14].

Figure 1. Mechanical fluid-structure coupling

Nowadays, the numerical approach tend to propose general coupling algorithms with specific solvers for each sub-problems, with various coupling strategies [4] [7] [15] [17], depending on the physical coupling phenomenon.

As the numerical design naval propulsion structures needs taking fluid-structure interaction into account, DCN Propulsion launched an R&D study in order to apply numerical methods on industrial projects. From the academic point of view, the coupling process seems not to pose any particular difficulties. From the industrial point of view, an investigation of the various aspects of the numerical fluid-structure coupling on a simple case is necessary before considering a full-scale coupling with existing commercial codes for industrial structure design.

The present study focuses on implicit and explicit coupling procedures for non-linear fluid-structure problems with the view to comparing the two approaches from numerical accuracy and stability points of view.

The case under study is first presented: explicit and implicit coupling procedures are then exposed. Numerical results are discussed for a linear single degree of freedom and for the non-linear multi-degrees of freedom problems.

2. A NON-LINEAR FLUID-STRUCTURE PROBLEM

2.1. Problem geometry and dynamic equation

The fluid-structure coupled problem studied in the present paper is described by Fig. 2¹ below.

Figure 2. 1D structure-3D fluid non-linear coupled problem

The structure problem is described by the following beam non-linear equation of motion, taking into account geometrical non-linearity [10] [18]. The coupled traction/bending equation of motion are formulated in the relative frame as:

$$\rho_s S \frac{\partial^2 u}{\partial t^2} + \frac{\partial}{\partial z} \left\{ ES \left[\frac{\partial u}{\partial z} + \frac{1}{2} \left(\frac{\partial v}{\partial z} \right)^2 \right] \right\} = 0 \quad (1)$$

$$\rho_s S \frac{\partial^2 v}{\partial t^2} + \frac{\partial^2}{\partial z^2} \left(EI \frac{\partial^2 v}{\partial z^2} \right) - \frac{\partial}{\partial z} \left\{ ES \left[\frac{\partial u}{\partial z} + \frac{1}{2} \left(\frac{\partial v}{\partial z} \right)^2 \right] \frac{\partial v}{\partial z} \right\} = -\rho_s S \gamma \quad (2)$$

The boundary conditions are:

$$u|_{z=0} = 0 \quad v|_{z=0} = 0 \quad \frac{\partial v}{\partial z}|_{z=0} = 0 \quad (3)$$

for the beam clamped end, and:

$$\left(\frac{\partial u}{\partial z} + \frac{1}{2} \left(\frac{\partial v}{\partial z} \right)^2 \right) \Big|_{z=L} = 0 \quad \frac{\partial^2 v}{\partial z^2} \Big|_{z=L} = 0 \quad \frac{\partial^3 v}{\partial z^3} \Big|_{z=L} = 0 \quad (4)$$

for the beam free end.

The fluid problem is described by the general NAVIER-STOKES equations. The conservation equations are integrated over a moving control volume $\Omega(t)$ of boundary $\partial\Omega(t)$, using the LIEBNITZ rule and the GAUSS theorem. This leads to the global conservation equation over an arbitrary control volume [8] in ALE (Arbitrary Lagrangian Eulerian) formulation [16]:

- Mass conservation

$$\frac{d}{dt} \left(\int_{\Omega(t)} \rho d\Omega \right) + \int_{\partial\Omega(t)} \rho (v_j - v_j^*) dn_j = \int_{\Omega(t)} S_\rho d\Omega \quad (5)$$

- Momentum conservation

$$\begin{aligned} \frac{d}{dt} \left(\int_{\Omega(t)} \rho v_i d\Omega \right) + \int_{\partial\Omega(t)} \rho v_i (v_j - v_j^*) dn_j = \\ - \int_{\partial\Omega(t)} p dn_j + \int_{\partial\Omega(t)} \mu \left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right) dn_j + \int_{\Omega(t)} S_{v_i} d\Omega \end{aligned} \quad (6)$$

In Eqs. (5) and (6), S_ρ and S_{v_i} stand respectively for mass and momentum sources: the fluid problem can then be formulated in the moving frame. The fluid unknowns are the pressure and velocity fields p, v_i , but as the conservation equations are integrated over a moving control volume, Eqs. (5) and (6) show another unknown fluid that is the grid velocity v_j^* : another equation has to be used to close the fluid problem. This supplementary equation is the space law conservation [5], which is written:

$$\frac{d}{dt} \left(\int_{\Omega(t)} d\Omega \right) = \int_{\partial\Omega(t)} v_j^* dn_j \quad (7)$$

The fluid problem is then fully characterized by Eqs. (5) to (7)². Additional boundary conditions are to be taken into account such as imposed pressure or velocity. The coupling

with the structure problem is expressed as $\frac{\partial \mathbf{u}}{\partial t} \Big|_{\Gamma} = \mathbf{v} \Big|_{\Gamma}$ where

$\mathbf{u}$ is the structure displacement and Γ is the fluid-structure interface.

¹ It is an extension in the 3D case of the non-linear 2D problem studied in a previous publication, this elementary case been studied with an explicit coupling procedure [18]. The present study aims at using the implicit procedures developed by Abouri & al. for a fluid-rigid body system [1].

² In the case of a compressible fluid, the fluid state law $\rho(p)$ allows a closure of the problem.

2.2. Discretization of the structure problem: finite element

Space discretization of the structure problem is carried out using two nodes beam finite element with linear shape functions for traction-compression u and cubic shape functions for flexion v (see Fig. 3)

Figure 3. Beam finite element with two nodes, three degrees of freedom per node

Finite element discretization procedure leads to the following matrix system [18]:

$$\begin{bmatrix} M_u & 0 \\ 0 & M_v \end{bmatrix} \begin{Bmatrix} \ddot{U} \\ \ddot{V} \end{Bmatrix} + \begin{bmatrix} K_u & 0 \\ 0 & K_v \end{bmatrix} \begin{Bmatrix} U \\ V \end{Bmatrix} + \begin{Bmatrix} R_u(V) \\ R_v(U) \end{Bmatrix} = \begin{Bmatrix} 0 \\ \Phi_v \end{Bmatrix} \quad (8)$$

where the mass matrices are given by $[M_u] = \int_0^L \rho_s S \{N_u\} \{N_u\}$

and $[M_v] = \int_0^L \rho_s S \{N_v\} \{N_v\}$. the stiffness matrices are

expressed as $[K_u] = \int_0^L ES \left\{ \frac{\partial N_u}{\partial z} \right\} \left\{ \frac{\partial N_u}{\partial z} \right\}$ and

$[K_v] = \int_0^L EI \left\{ \frac{\partial^2 N_v}{\partial z^2} \right\} \left\{ \frac{\partial^2 N_v}{\partial z^2} \right\}$. and the loading vector is written

$\{\Phi_v\} = - \int_0^L \rho_s S \gamma \{N_v\} + \Phi$ where Φ is the projection of fluid

forces onto the structure finite element mesh (see next subsection).

The non-linear terms are:

$$\{R_u(V)\} = \int_0^L \frac{ES}{2} \left\{ \frac{\partial N_u}{\partial z} \right\} \{V\} \left\{ \frac{\partial N_v}{\partial z} \right\} \left\{ \frac{\partial N_v}{\partial z} \right\} \{V\} \quad (9)$$

$$\begin{aligned} \{R_v(U, V)\} = & \int_0^L ES \left\{ \frac{\partial N_v}{\partial z} \right\} \left[\left\{ \frac{\partial N_u}{\partial z} \right\} \{U\} + \right. \\ & \left. \frac{1}{2} \{V\} \left\{ \frac{\partial N_v}{\partial z} \right\} \left\{ \frac{\partial N_v}{\partial z} \right\} \{V\} \right] \left\{ \frac{\partial N_v}{\partial z} \right\} \{V\} \end{aligned} \quad (10)$$

Mass and stiffness matrices are calculated analytically and non-linear terms are evaluated with numerical procedures.

2.3. Discretization of the fluid problem: finite volume

The general form of an integrated conservation equation for a fluid unknown ϕ is the following one:

$$\frac{d}{dt} \left(\int_{\Omega(t)} \rho \phi d\Omega \right) + \int_{\partial\Omega(t)} \rho (v_j - v_j^*) \phi dn_j = \int_{\Omega(t)} S_\phi d\Omega + \int_{\partial\Omega(t)} \Gamma \frac{\partial \phi}{\partial x_j} dn_j \quad (11)$$

Equation (11) is discretized using a finite volume method. The fluid domain is divided in elementary fluid cells. for which an integrated conservation equation is written. Figure 4 gives a typical representation of a computational grid for a 2D Cartesian problem.

A moving mesh technique is used: the elementary grid velocity is deduced from known node displacement in order to satisfy Eq. (7) [8]. The grid velocity v_i^* appearing in the right side of Eq. (7) becomes then a source term on the left-hand side. Integrated conservation equations are written for each cell: P is the cell center. $WW, W, SW, S, SE, E, EE, NE, N$ are the centers of neighboring cells.

Figure 4. 2D fluid computational grid

The time dependant in Eq. (11) are approximated by an EULER scheme. *i.e.* writing:

$$\frac{d}{dt} \left(\int_{\Omega_P(t)} \rho \phi d\Omega \right) \approx \frac{\rho \Omega}{\delta t} (\phi_P^{n+1} - \phi_P^n) \quad (12)$$

The convective and diffusive terms are calculated using finite difference approximation. leading to the global expression [8]:

$$\int_{\Omega_P(t)} \rho v_j \phi - \Gamma \frac{\partial \phi}{\partial x_j} d\Omega_j \approx \sum_M C_M^{n+1} \phi_M^{n+1} - D_M^{n+1} \phi_M^{n+1} \quad (13)$$

where M stands for the neighboring points of cell Ω_P .

Taking into account Eqs. (12) and (13) leads to the following algebraic non-linear system $[A(\phi)]\phi = \{b\}$, that is:

$$A_P \phi_P^{n+1} = \sum_M A_P \phi_M^{n+1} + b_P \phi_P^n \quad (14)$$

From discretization practices presented above, it is possible to produce the following algebraic system resulting from the finite volume discretization of the momentum equation and POISSON equations:

$$A_P v_{i,p}^{n+1} = H(v_i^{n+1}) + D(p^{n+1}) + S_{v_i}^{n+1} \quad (15)$$

$$A_P p_P^{n+1} = G(p^{n+1}) + S_p(v_i^{n+1}) \quad (16)$$

Equations (15) and (16) couple the pressure and velocity unknowns. An implicit iterative scheme is used to calculate the new values of the velocity and pressure field [11]. The algorithm starts with an estimation of the velocity field, by solving the following non-linear system, obtained from Eq. (15) by using the pressure field computed at the previous time step:

$$A_P v_{i,p}^{n+1,(0)} = H(v_i^{n+1,(0)}) + D(p^n) + S_{v_i}^{n+1} \quad (17)$$

Equations (15) and (16) are then used to produce successive correction, by using the iterative relations:

$$\begin{cases} A_P v_{i,p}^{n+1,(q+1)} = H(v_i^{n+1,(q)}) + D(p^{n+1,(q)}) + S_{v_i}^{n+1} \\ A_P p_P^{n+1,(q)} = G(p^{n+1,(q)}) + S_p(v_i^{n+1,(q)}) \end{cases} \quad (18)$$

The iterative scheme is stopped when a convergence criterion is satisfied or when a maximum number of inner iterations is reached.

From the practical point of view, the coupled problem is solved using the CFD code Star-CD [22], which allows the numerical calculation for the fluid problem part. The structure problem part is solved using FORTRAN subroutines developed within the CFD code.

3. COUPLING PROCEDURE

The key of the numerical method for the coupled problem lays in the coupling procedure between the fluid and structure codes.

The coupling procedure allows coupling the problems:

- ✓ in space:
- ✓ in time.

3.1. Coupling in space

The space coupling procedure handles numerical exchanges between the fluid and structure problems. Structure displacement are transferred from the structure mesh to the fluid mesh, and fluid forces are projected from fluid boundaries to structure mesh, as sketched by Fig. 5.

Figure 5. FE structure code and FV fluid code space coupling

As fluid and structure meshes can be very different (a few finite element are enough to describe the structure dynamic, an important number of fluid finite volume are required to solve the fluid problem, with local refinement to describe boundary layer), the space coupling procedure uses interpolation techniques in the displacement/force exchanges between the two sub-problems [2].

Since the fluid problem is re-meshed at each time step to take into account structure deformation, the fluid finite volume can undergo great shape deformation. Many re-meshing coupling techniques can be developed to preserve the fluid mesh quality [19]. As the fluid geometry is rather simple in the present case, it is possible to develop a re-meshing procedure that produces little deformation of the fluid cells, based on a purely geometrical approach of the problem.

Figure 6 shows a 2D half slice of the fluid mesh, in the initial state and a deformed state: fluid cells length ratios are preserved even for large amplitude inner cylinder motion³.

³ For very large displacement, i.e. when the inner cylinder almost comes into contact with the outer one, the proposed re-meshing technique will fail (other ones too !...) to preserve mesh quality. In this case, a numerical technique based on birth/death fluid cells approach would be useful. Such a method is available in the STAR-CD code [22] and is currently investigated in the present coupled problem.

Initial mesh Deformed mesh
Figure 6. Initial and deformed mesh

3.2. Coupling in time

Coupling in time can be carried out using either explicit or implicit approaches. In a previous paper, the elementary case described by Fig. 1 has been studied with explicit coupling procedure (see SIGRIST *et al.* [18]). The purpose of the present study is to apply implicit coupling procedure, using the fluid-rigid body implicit method developed by ABOURI *et al.* [1].

A rigid-body movement is described by a set of ordinary differential equations of the form:

$$\begin{cases} m \frac{d\dot{\mathbf{X}}}{dt} = \mathbf{F}(\dot{\mathbf{X}}, \mathbf{x}, t) \\ \frac{d\mathbf{X}}{dt} = \dot{\mathbf{X}} \end{cases} \quad (19)$$

where $\mathbf{X}$ and $\dot{\mathbf{X}}$ are the displacement and velocity of the solid system.

Equation (19) can be solved using an explicit step by step method. A first order numerical scheme is for instance:

$$\dot{\mathbf{X}}_{n+1} = \dot{\mathbf{X}}_n + dt \frac{\mathbf{F}(\dot{\mathbf{X}}_n, \mathbf{x}_n, t_n)}{m} \quad (20)$$

Such a method has been previously used to perform numerical simulation of the generic coupled case [18]. The explicit approach is rather straightforward, in particular with non-linear problems.

However, stability problems are likely to occur while using explicit schemes. Implicit time discretization can be used to overcome such difficulty. A first order implicit approximation of Eq. (19) reads:

$$\dot{\mathbf{X}}_{n+1} = \dot{\mathbf{X}}_n + \frac{dt}{2} \left(\frac{\mathbf{F}(\dot{\mathbf{X}}_n, \mathbf{x}_n, t_n) + \mathbf{F}(\dot{\mathbf{X}}_{n+1}, \mathbf{x}_{n+1}, t_{n+1})}{m} \right) \quad (21)$$

The following non-linear system has then to be solved to calculate $\dot{\mathbf{X}}_{n+1}$:

$$\psi(\dot{\mathbf{X}}_{n+1}) = v_{n+1} - v_n - \frac{dt}{2} \left(\frac{\mathbf{F}(\dot{\mathbf{X}}_n, \mathbf{x}_n, t_n) + \mathbf{F}(\dot{\mathbf{X}}_{n+1}, \mathbf{x}_{n+1}, t_{n+1})}{m} \right) = 0 \quad (22)$$

As it is well known, in order to accelerate convergence the most widely used and most robust method for the solution of non-linear system is Newton-Raphson method. It requires the evaluation of derivatives that are approximated by finite differences. Solving $\psi(\dot{\mathbf{X}}_{n+1}) = 0$ is performed with the following iterations:

$$\dot{\mathbf{X}}_{n+1}^{(k)} = \dot{\mathbf{X}}_{n+1}^{(k-1)} - \left[\nabla \psi(\dot{\mathbf{X}}_{n+1}^{(k-1)}) \right]^{-1} \psi(\dot{\mathbf{X}}_{n+1}^{(k-1)}) \quad (23)$$

The iterative procedure starts with the initial value $\dot{\mathbf{X}}_{n+1}^0$, which can be predicted by an explicit scheme of the same order or simply from the result of the last time step ($\dot{\mathbf{X}}_{n+1}^0 = \dot{\mathbf{X}}_n$).

In the studied case, the structure equation – see Eq. (8) – is written:

$$\mathbf{M}\ddot{\mathbf{X}} + \mathbf{K}(\mathbf{X})\mathbf{X} = -\mathbf{M}\mathbf{D}\gamma + \Phi \quad (24)$$

where γ is the imposed shock in direction $\mathbf{D}$ and Φ is the fluid force. Implicit discretization of Eq. (24) also leads to a non-linear problem $\psi(\dot{\mathbf{X}}_{n+1}) = 0$, which is solved with the algorithm described by Fig. 7 below. At each time step, the procedure proceeds as follows.

1. Initialisation of structure displacement $\mathbf{X}_{n+1}^0 = \mathbf{X}_n$ and velocity $\dot{\mathbf{X}}_{n+1}^0 = \dot{\mathbf{X}}_n$, of structure pressure $p_{n+1}^0 = p_n$ and velocity $v_{n+1}^0 = v_n$, of fluid mesh $\Omega_{n+1}^0 = \Omega_n$ and fluid forces $\Phi_{n+1}^0 = \Phi_n$.

2. Structure calculation (with finite element discretization) by solving Eq. (23) to obtain $\dot{\mathbf{X}}_{n+1}^{(k)}$ and then $\mathbf{X}_{n+1}^{(k)}$.

3. Coupling in space to fluid problem. Fluid mesh $\Omega_{n+1}^{(k)}$ and fluid boundary condition at $\Gamma_{n+1}^{(k)}$ are deduced from structure displacement and velocity.

4. Fluid calculation (with finite volume discretization) by solving Eq. (14) with the PISO algorithm.

5. Coupling in space to structure problem. Fluid forces acting on $\Phi_{n+1}^{(k)}$ are projected to the structure mesh.

6. **Coupling in time** between the two problems by performing a convergence test on structure displacement $\|X_{n+1}^k - X_{n+1}^{k-1}\| \leq \varepsilon$. If the convergence criterion is satisfied, the coupling loop ends; otherwise, calculation proceeds within the coupling loop.

Coupling iteration are performed with the same time step we used a restart technique for moving mesh problems automated with an operating system script.

When convergence criterion is not taken into account, the algorithm is equivalent to an explicit coupled procedure. Explicit and implicit approaches can then be easily compared.

Figure 7. Explicit and implicit coupling method

4. APPLICATION EXAMPLES

Explicit and implicit coupling procedures are tested on two application examples.

4.1. Fluid-cylinder coupled problem

First application is a fluid-rigid problem, as defined by Fig. 8. Free oscillations of the rigid cylinder are given by $u(t) = u_0 \cos(2\pi f_h t)$ with $f = \frac{1}{2\pi} \times \sqrt{\frac{k}{m + m_h}}$ where m_h is the fluid added mass [9].

Figure 8. Rigid cylinder oscillating in a confined non viscous fluid

Table 1 gives the numerical value used for numerical simulations.

Parameter	Value
R [m]	0.1
R' [m]	0.2
h [m]	0.01
k [N/m]	60
m [kg]	1
m _b [kg]	0.52
f ₀ [hz]	1.233
f _b [hz]	1.000

Table 1. Fluid-cylinder problem

Figure 9 gives the numerical results for the cylinder free oscillations and compares the analytical solution to the numerical solution obtained with the explicit method for time steps $dt = 0.01$ s and $dt = 0.001$ s and with the implicit method for time step $dt = 0.01$ s.

Figure 10 gives the number of inner coupling iterations during the simulation.

These results illustrate the numerical properties of explicit and implicit procedures:

- ✓ explicit coupling introduces a numerical damping that can be decreased using a smaller time steps.
- ✓ implicit coupling is free of numerical damping even when large time steps are used. Although implicit coupling with $dt = 0.01$ s requires an average number of inner iteration around 5, computation time is still lower than in the case

of explicit coupling with $dt = 0.001$ s. for equivalent results.

Figure 9. Free oscillation calculation. Cylinder displacement

Figure 10. Free oscillation calculation. Coupling iterations

4.2. Fluid-beam coupled problem

Second application is the fluid-beam coupled problem defined by Fig. 2. The whole system is subjected to a sine wave acceleration:

$$\gamma(t) = \begin{cases} \gamma \times \sin\left(2\pi \frac{t}{\tau}\right) & \text{if } 0 \leq t \leq \tau \\ 0 & \text{if } \tau < t \end{cases} \quad (25)$$

Geometry and physical parameters of the problem are given in Tab. 2.

Calculations are performed using explicit and implicit algorithms.

Parameter	Value
R [m]	0.1
L [m]	1
e [m]	0.02
ρ_s [kg m ³]	7800
E [Pa]	$6.04 \cdot 10^8$
R' [m]	0.2
H [m]	1
ρ_f [kg m ³]	1000
τ [s]	0.1
γ [m s ⁻²]	5

Table 2. Fluid-beam problem

Figure 11 gives the relative displacement (in the moving frame) of the beam free end versus time in various conditions. Figure 12 gives the corresponding absolute acceleration. The numerical results are compared to a finite element computation, which is the reference case [18].

Figure 11. Shock calculation. Beam relative displacement

Figure 12. Shock calculation. Beam absolute acceleration

Numerical results obtained with implicit coupling for $dt = 0.001$ s are in good agreement with the finite element calculation. both in terms of displacement and acceleration.

The procedure is slightly numerically dissipative but remain numerically stable.

Computation with explicit coupling with the same time step seem to give equivalent results in terms of displacement, but the procedure is more dissipative and potentially unstable, as can be shown by acceleration calculation.

Numerical dissipation could be expected to be lowered in explicit coupling by using a smaller time step ($dt = 0.0001$ s) but this increases numerical instability.

Figure 13 gives the number of coupling iterations in the case of implicit coupling. The average number is around 3, the corresponding required computational time effort is by far compensated by the use of larger time steps.

Figure 13. Shock calculation. Coupling iterations

These numerical results are a good illustration of the numerical properties of implicit coupling for non-linear fluid-structure coupled simulations.

5. CONCLUSION

An elementary study of coupled fluid-structure simulations by finite element-finite volume coupling using standard numerical tools has been proposed in the present paper.

The study focuses on time coupling procedures. Explicit and implicit approaches are applied in a generic non-linear fluid-structure coupled test case, using a CFD code.

Implicit and explicit procedure are first recalled. Various calculation results are then exposed and compared from the numerical points of view: dissipation and stability properties of the time coupling schemes are illustrated.

Industrial applications of the coupled techniques are currently under investigation in order to study flow induced vibrations

phenomenon in tube bundle (following the studies of LONGATTE & SOULI [3]).

REFERENCES

- [1] D. ABOURI, A. PARRY, A. HAMDOUNI, *A Stable Fluid Rigid Body Interaction Algorithm Application to Industrial Problems*, Pressure Vessel and Piping, San Diego, 25-29 July 2004.
- [2] A. BECKER, Coupling Fluid (CFD) and Structural (FE) models Using Finite Interpolation Elements *Aerospace Science and technology*, **4** (1), 13-22, 2000.
- [3] Z. BENDJEDOU, E. LONGATE, A. ADOBES, M. SOULI, *Steam Generator Tube Vibrations Experimental determination versus ALE Computation of Fluidelastic Forces*, Pressure Vessel and Piping, Cleveland, 20-24 July 2003.
- [4] F. CASADEI, J.P. HALLEUX, A. SALA, F. CHILLE, Transient Fluid Structure Interaction Algorithm for Large Industrial Applications, *Computer Methods in Applied Mechanical Engineering*, **190**, 3081-3110, 2001.
- [5] I. DEMIRDZIC, M. PERIC, Space Conservation law in Finite Volume Calculations of Fluid Flow, *International Journal for Numerical Methods in Fluids*, **8**, 1037-1050, 1988.
- [6] M.A. DOKAINISH, K. SUBBARAJ, A Survey of Direct Time-Integration Methods in Computational Dynamics-I: Explicit Methods, *Computer & Structures*, **32**, 1371-1386, 1989.
- [7] C. FARHAT, M. LESOINNE, N. MAMAN, Mixed Explicit/Implicit Time Integration of Coupled Aeroelastic Problems, *International Journal of Numerical Methods in Fluids*, **21**, 807-835, 1995.
- [8] J.H. FERZIGER, M. PERIC, *Computational Methods for Fluid Dynamics*, Springer-Verlag, 1999.
- [9] R.J. FRITZ, The Effect of Liquids on the Dynamic Motion of Immersed Solids, *Journal of Engineering for Industry*, 167-173, 1972.
- [10] J. HINO, Y. YOSHIMURA, N. ANANTHANARAYANA, Vibration Analysis of Non-Linear Beams Subjected to a Moving Load Using the Finite Element Method, *Journal of Sound and Vibration*, **100**, 477-491, 1985.
- [11] R.I. ISSA, Solution of The Implicitly Discretised Fluid Flow Equation by Operator Splitting, *Journal of Computational Physics*, **62**, 40-65, 1985.
- [12] E. DE LANGRE, *Fluides et Solides*, Les éditions de l'Ecole Polytechnique, 2002.
- [13] J. MAKERLE, Fluid-Structure Interaction Problems, Finite Element Approach and Boundary Elements Approaches, A Bibliography, *Finite Elements in Analysis and Design*, **31**, 231-240, 1999.
- [14] H. J.P. MORAND, R. OHAYON, *Fluid Structure Interaction*, Wiley & Sons, 1995.
- [15] J.M.T. PENROSE, C.J. STAPLES, Implicit Fluid-Structure Coupling for Simulation of Cardiovascular Problems, *International Journal for Numerical Methods in Fluids*, **1**, 1-13, 2001.
- [16] J. SARRATE, A. HUERTA, J. DONEA, *Arbitrary Lagrangian Eulerian Formulation for Fluid Multi-Rigid Bodies Interaction Problems*, Computation Mechanics, New Tread and Applications, 1998.
- [17] M. SCHÄFFER, *Coupled Fluid-Solid Problems Survey on Numerical Approaches and Applications*, Pressure Vessel and Piping, Cleveland, 20-24 July 2003.
- [18] J.F. SIGRIST, C. LAINE, *Numerical Simulation of a Non-Linear Coupled Fluid-Structure Problem by Explicit Finite Element Finite Volume Coupling*, Pressure Vessel and Piping, Denver, 17-21 July 2005.
- [19] R.M.C. SO, Y. LIU, Y.G. LAL, *Mach Shape Preservation for Flow-Induced Vibrations Problems*, International Mechanical Engineering Congress and Exposition, New Orleans, 17-22 November 2002.
- [20] K. SUBBARAJ, M.A. DOKAINISH, A Survey of Direct Time-Integration Methods in Computational Dynamics-II: Implicit Methods, *Computer & Structures*, **32**, 1387-1401, 1989.
- [21] Z.U.A. WARSI, Conservation form of the Navier Stokes Equations in General Non Steady Coordinates, *American Institute of Aeronautics and Astronautics Journal*, **19**, 240-242, 1980.
- [22] STAR-CD Methodology-User Guide, CD Adapco Group, 2003.