A Fully-Implicit Multigrid Driven Algorithm for Time-Resolved Non-linear Free-Surface Flow on Unstructured Grids

by

B. Liou, L. Martinelli, and A. Jameson

Department of Mechanical and Aerospace Engineering
Princeton University, Princeton, New Jersey, USA

Proceedings
1996 ASME Fluid Engineering Division Summer Meeting
San Diego

July 1996

A Fully-Implicit Multigrid Driven Algorithm for Time-Resolved Non-linear Free-Surface Flow on Unstructured Grids

Biing-Horng Liou, Luigi Martinelli and Antony Jameson

Department of Mechanical and Aerospace Engineering
Princeton University
Princeton N.J. 08544
U.S.A.

Abstract
A fully-implicit multigrid method has been developed for the computation of time dependent free surface flow. The method uses the generalized artificial compressibility approach to couple the incompressible Euler equations and continuity equation in a hyperbolic manner which allows several techniques for convergence acceleration to be implemented. Two grid adaption strategies are used to capture wave motions as well as wavemaker's oscillations. Comparisons are made between these two techniques and experimental data of the flow field around a NACA 0012 airfoil placed beneath an initially calm free surface. Transient solutions of a wavemaker oscillating horizontally are also presented to validate the new method.

Introduction
For decades, considerable work has been directed towards the study of nonlinear free surface flows not only to gain new insights into the problem but also because of the practical importance in ship hydrodynamics. However, even with the new generation of computers, two major difficulties still remain which make the development of efficient and accurate free surface flow solvers a challenging task.

One difficulty is the continuous evolution of the physical domain caused by the free surface deformation. Thus the need to adapt the mesh is inevitable. There are two viable approaches. The first one makes use of a kind of spring mechanism or transformation to shift the grids [8, 19]. However, this method fails when the waves start to overturn. Moreover, the numerical discretization on such meshes results into inaccuracies because the approximation does not take into account mesh-point rearrangement and grid distortion. This problem is prevented by the second approach which deletes and inserts points while the simulation progresses. Both methods are implemented in our work and the results are compared.

The other major difficulty, which is common to any incompressible flows not just free surface flows, is the enforcement of a solenoidal velocity field, as it is required by the continuity equation. Lack of a pressure evolution term precludes the straightforward application of efficient time-marching algorithms which are available for hyperbolic problems. One of the approaches to circumvent this difficulty is to introduce a pseudo-temporal evolution term for the pressure in the continuity equation as in the well-known artificial compressibility method of Chorin [9]. This transforms the governing equations into a hyperbolic system at the expense of time accuracy. However, the time accuracy can be recovered by augmenting spatial residuals in the momentum equations with the discretized time derivatives, and by driving these modified residuals to zero at each mesh point and every time step. This generalized artificial compressibility approach ensures a direct coupling between the velocity and pressure fields upon convergence of the *pseudo-transient* at each time step. The use of this strategy for incompressible flow has been originally proposed by Rogers and Kwak [15]. A similar approach has also been used by Miyaka et al. [16] using an explicit, up to a second order accurate discretization in time, rational Runge-Kutta scheme for the subiterations. A very efficient method, which couples a second order accurate backward differencing of the temporal derivatives and a very efficient finite-volume multigrid strategy, has been described and validated in [1, 12, 13] for both two-dimensional Euler and Navier-Stokes equations on structured quadrilateral meshes. The method has been also implemented and validated for the solution on unstructured grids of both incompressible and compressible flow on oscillating airfoil by Lin [6]. In the present algorithm, fast convergence to a steady state of the pseudo-transient is achieved by making use of a multigrid technique originally developed by Jameson for compressible flow [1], and adapted by Hino, Martinelli and Jameson [8] for steady free-surface calculations on triangular grids. Details on the space discretization can be found in reference [5] and a comprehensive study of the artificial compressibility method for unstructured grids is given in [10]. The A-stable discretization in time allows the stability constraint on the physical time step to be relaxed, while standard convergence acceleration techniques such as local pseudo-time stepping and residual averaging are applied to the pseudo-transient iteration. Also, to alleviate the stiffness effects stemming from the unsteady source terms included in the residuals, a point-implicit five-stage Runge-Kutta scheme is constructed following the guidelines given in [4]. Finally, the range of the characteristic wave speeds associated with the hyperbolic pseudotransient problem is optimized for better convergence by employing a suitable form of the local preconditioning [11, 13].

Discretization of the Governing Equations

Governing Equations

Consider a general two-dimensional homogeneous incompressible inviscid free surface flow problem. Let the reference length, velocity and density be L_0, U_0 and ρ_0. The dimensionless Cartesian velocity components and dimensionless pressure are denoted by u, v and $\hat{p}$ respectively. Let $p=\hat{p}+\frac{1}{F^2}$ be the pressure minus the hydrostatic part, where the Froude number $F=\frac{U_0}{\sqrt{gh}}$, h is height of the free surface, and g is the gravitational acceleration. The dimensionless governing equations which consist of the continuity equation and the time-dependent momentum equation are

$$\frac{d}{dt}\int_{V(t)} dV + \int_{S(t)} (\mathbf{u}_r \cdot \mathbf{n})\, dS = 0 \qquad (1)$$

and

$$\frac{d}{dt}\int_{V(t)} \mathbf{u} dV + \int_{S(t)} \mathbf{u}(\mathbf{u}_r \cdot \mathbf{n})\, dS = -\int_{S(t)} p\mathbf{n} dS \qquad (2)$$

where $\mathbf{u}$ is the velocity measured with respect to an inertial reference frame , $\mathbf{u}_r=\mathbf{u}-\mathbf{u}_b$ is the velocity of the fluid relative to the control surface with velocity $\mathbf{u}_b$, and $\mathbf{n}$ is the unit normal. Note that the time rate of volume change in equation (1) is of importance. In semi-discrete form, equations (1) and (2) become

$$\frac{d}{dt}[\mathbf{T}_{ij}V_{ij}] + \mathbf{R}(\mathbf{w}_{ij}) = 0$$

where $\mathbf{T}_{ij}=\mathbf{w}_{ij}\cdot\mathbf{I}^m+\mathbf{K}$, $\mathbf{I}^m=diag[0,1,1]$ is the modified identity, $\mathbf{K}=[1,0,0]$, $\mathbf{w}_{ij}=[p,u,v]$ and residual $\mathbf{R}(\mathbf{w}_{ij})$ is obtained by approximating convective fluxes with central difference in space plus a third order artificial dissipation term to prevent an odd-even decoupling. A backward difference discretization in time considered here is of the form

$$\frac{d}{dt}=\frac{1}{\Delta t}\sum_{q=1}^{k}\frac{1}{q}[\Delta^-]^q, \Delta^-=(\cdot)^{n+1}-(\cdot)^n$$

In particular, dropping the subscripts i, j for clarity, for a second order discretization in time , one obtains

$$\mathbf{R}^*(\mathbf{w}) = \frac{1}{\Delta t}(q_1\mathbf{T}V + q_2\mathbf{T}^nV^n + q_3\mathbf{T}^{n-1}V^{n-1}) + \mathbf{R}(\mathbf{w}) = 0$$

where $\mathbf{q}=(3/2,-2,1/2)$ and $\mathbf{R}^*(\mathbf{w})$ is the augmented residual. In order to perform time resolved calculation, generalized artificial compressibility approach is used, which results in a system of coupled O.D.E.'s to be solved to convergence at each time step

$$\frac{d\mathbf{w}V}{dt^*} + \mathbf{P}_r\cdot\mathbf{R}^*(\mathbf{w}) = 0 \qquad (3)$$

where the preconditioning matrix is $\mathbf{P}_r=diag[\Gamma^2,1,1]$ and $\Gamma^2=\max(0.25, u^2+v^2)$. A point-implicit, k-stage Runge-Kutta method which can be cast as

$$\begin{aligned}
\mathbf{w}^{(0)} &= \mathbf{w}^n \\
&\cdots \\
\mathbf{w}^{(i)}V^{n+1}(1+\alpha_i q_1\frac{\Delta t^*}{\Delta t}\mathbf{I}^m) &= \mathbf{w}^{(i-1)}V^{n+1} \\
-\alpha_i\Delta t^*[\mathbf{R}^{*(i-1)} &- \frac{1}{\Delta t}q_1\mathbf{w}^{(i-1)}\cdot\mathbf{I}^mV^{n+1}] \\
&\cdots \\
\mathbf{w}^{n+1} &= \mathbf{w}^{(k)}
\end{aligned}$$

where $\mathbf{w}^n$ is the value of $\mathbf{w}$ after n pseudotime steps, is applied to drive equation (3) to steady state in pseudotime t^*. Once a "steady-state" is reach, $\mathbf{R}^*(\mathbf{w})=0$ is satisfied and one step in real time has advanced.

Several efficient techniques are employed to accelerate convergence at each time step. The most important one is the multigrid scheme, which also uses separately generated meshes. The details of the multigrid scheme can be found in [5, 7]. Another is called the local time stepping technique which allows each control volume to be advanced in pseudo time by its own maximum local pseudo time step. Residual averaging is also an effective method to increase the pseudo time step by collecting information from residuals at neighboring points [7]. These techniques are performed on the subiterations and do not affect time accuracy. Note that equation (3) can also be used to obtain a steady state solution when Δt approaches infinity, and $\mathbf{R}^*(\mathbf{w}) = \mathbf{R}(\mathbf{w})$. In this paper both steady and unsteady flow problems will be examined.

Boundary Conditions

A free-slip condition on the solid bodies is implemented as

$$\mathbf{u}_r \cdot \mathbf{n} = 0 \tag{4}$$

which states that there is no normal flow through the bodies.

The free surface condition consists a dynamic and a kinematic condition. The former states the continuity of normal stress on the air-liquid surface. For an inviscid flow, and neglecting the surface tension, this is expressed as

$$\hat{p} = p_0, \quad \text{at} \quad y = h$$

or

$$p = p_0 + \frac{h}{F^2}, \quad \text{at} \quad y = h \tag{5}$$

where p_0 is the atmospheric pressure which is assumed to be constant. The latter condition describes that the free surface is a material surface and can be written as

$$\frac{\partial h}{\partial t} + u_r \frac{\partial h}{\partial x} - v = 0, \quad \text{at} \quad y = h \tag{6}$$

This time-marching equation for the wave height must be solved together with the bulk flow equations. Thus, a pseudotime derivative term must be added to implement the dual time stepping method described earlier.

For the steady flow calculations, additional boundary conditions are needed at the far field boundaries because of the truncation of the computational domain. Thus, a wave dumper is imposed on equation (6) near the downstream numerical boundary to prevent reflection of waves to the domain as described in [8, 20]. A uniform flow with an undisturbed free surface is imposed at the inflow: $u=1, v=0, p=0, h=0$. Also at the bottom, a deep water approximation is applied and the pressure is set to the unperturbed value $p=0$. At boundary nodes, all the other flow quantities are obtained by using one-sided control volumes.

Grid Generation And Grid Adaption

A computational mesh is necessary to discretize the governing equations in space. Ultimately, the quality of the mesh determines the accuracy of a numerical solution. Several methods have been proposed to generate unstructured meshes. Among them, a Delaunay triangulation technique is used in the present work. The Delaunay triangulation is defined such that no points lie inside the circumcircle of any triangle; it is easy to implement, and it is unique.

The method used to generate the mesh is based on an initial triangulation of the boundary points followed by the insertion of new points inside the domain according to a prescribed rule. Details on grid generation using Dealunay triangulation can be found in reference [14].

Two grid adaption techniques are used. One is the so called spring method which uses a spring mechanism to deform the mesh nodes. This method ignores remeshing procedure and therefore takes into account only mesh deformations. When the mesh distorts too much, the accuracy of the numerical solution will suffer. For this reason, another technique is implemented. In this second approach points are inserted in the domain when better resolution is required while existing points are deleted when they lie outside the domain. Thus, with this grid adaption strategy, the mesh quality can be guaranteed.

Results

Steady Flow Solver

We consider a uniform flow past a NACA0012 airfoil placed beneath the water. The characteristic length, velocity, and density are chosen as the chord length of the airfoil, the distant upstream velocity, and the density of the fluid respectively. The Froude number is 0.567 while the submergence c which is the distance between the center of the airfoil and the undisturbed free surface is either 1.034 or 0.951, as in the experiments of Duncan [17]. This problem is computed to assess both the efficiency and the accuracy of the multigrid solver, as well as to evaluate an alternative strategies of mesh rearrangement including the point insertion/deletion method described earlier. Such a remeshing strategy is more general than the spring method originally employed by Hino et al. [8] and may allow in the future to follow waves past the overturning.

Figure 1a shows the grid corresponding to the final converged solution for c=1.034, while figure 1b and 1c present comparisons of the wave elevations computed by using the two mesh movement strategies. The experimental data from reference [17] are also plotted. The computed waves are in phase with the one measured although there are slight difference in amplitude between the two. This indicates the level of accuracy that is achieved by our discretization method. The two adaption strategies perform equally well for the deeper submergence case, c=1.034. However, for the shallower one, c=0.951, the adaption method gives better accuracy than the spring method as it can be seen in figure 1c. This is due to the excessive mesh stretching exhibited by the spring method which impairs the accuracy of the computed solution for large amplitude waves. The convergence history plots of the root mean square error of the divergence of the velocity presented in figures 2a and 2b show that the multigrid scheme converges the velocity field to satisfy the continuity equation (solenoidality condition) to machine accuracy.

Unsteady Flow Solver

We consider a fluid in a finite rectangular tank with a vertical piston wavemaker at one end which corresponds to the experiment of [18]. The characteristic length, velocity, and density is set as the initial depth of the water H, $\sqrt{gH}$, and the fluid density. The nondimensional length of the tank is 20 in this study and the origin of the axis is fixed at the initial point of the intersection between wavemaker and unperturbed free surface. The horizontal velocity of the piston wavemaker $U(t)$ is prescribed as the following Fourier series

$$U(t) = \sum_{n=1}^{72} U_n cos(\omega_n t - \theta_n) \tag{7}$$

The amplitudes U_n, frequencies ω_n, and phase θ_n are tabulated in [18].

For this case it is found that 10-25 multigrid cycles are sufficient to make the velocity field solenoidal within a tolerance of 10^{-5} for each physical time step.

The free surface profiles at several different locations, i.e. x=3.17, 5.00, 6.67, 8.33, 10.00, and 12.17, are compared with linear theory. Figure 3 shows that our unsteady solver is able to capture the trend of the wave motion. Moreover, the computed solutions are in better agreement with the experimental data reported in the literature [18]. This is not surprising since we are accounting for most of the nonlinearity of the problem. More specifically, on a 296 × 16 mesh, when the computed wave height at x=3.17, t=25 is compared to the experiments, we find that:

	Δt	h
computation	0.10	-0.6239
computation	0.05	-0.6662
linear theory		-0.7197
experiments		-0.667

Thus, the error is less than 6% .

Conclusions

A fully-implicit multigrid solver has been developed for the solution of time resolved non-linear surface wave propagation problems. Good performance and accuracy is demonstrated in comparison with experimental evidence and linear theory. The proposed algorithm is quite flexible, and can be extended to both 3-Dimensional flows, and more complex viscous flow problem. Moreover, the method can be applied to problems which require the solution of the flow equations coupled with either a mathematical model of the structure and/or the motion of a body. Thus, we believe that this method will evolve into a viable tool for the simulation of non-linear "seakeaping" problems.

REFERENCES

[1] Jameson, A., "Time Dependent Calculations Using Multigrid, with Applications to Unsteady Flows Past Airfoils and Wings," AIAA Paper 91-1596, June 1991.

[2] Mitty, T. J., *Development of a Delaunay-Based Adaption Scheme with Applications to Complex Three-Dimensional Rotational Flows*, Ph.D. Thesis, Department of Mechanical and Aerospace Engineering, Princeton University, January 1993.

[3] Baker, T. J., "Automatic Mesh Generation for Complex Three-Dimensional Regions Using a Constrained Delaunay Triangulation," *Engineering with Computers*, Vol. 5, pp. 161–175, 1989.

[4] Melson, N. D., Sanetrik, M. D., and Atkins, H. L., "Time-Accurate Navier-Stokes Calculations with Multigrid Acceleration," Presented at the Sixth Copper Mountain Conference on Multigrid Methods, Copper Mountain, Colorado, April 1993.

[5] Mavriplis, D., *Solution of the Two-Dimensional Euler Equations on Unstructured Triangular Meshes*, Ph.D. Dissertation, Department of Mechanical and Aerospace Engineering, Princeton University, 1987.

[6] Lin, P.T., *Implicit Time Dependent Calculations for Compressible and Incompressible Flows on Unstructured Meshes* , Master Thesis, Department of Mechanical and Aerospace Engineering, Princeton University, 1994.

[7] Jameson, A., "Multigrid Algorithms for Compressible Flow Calculations," MAE Report 1743, October 1985, also in the Proceedings of the Second European Conference on Multigrid Methods, in Lecture Notes in Mathematics, Vol. 1228, edited by Trottenburg, U., and Hackbusch, W., Springer-Verlag, pp. 166–201, 1986.

[8] Hino, T., Martinelli, L., and Jameson, A., "A Finite-Volume Method with Unstructured Grid for Free Surface Flow Simulations," Sixth International Conference on Numerical Ship Hydrodynamics, Ship Research Institute, Tokyo, Japan, August 1993.

[9] Chorin, A. J., "A Numerical Method for Solving Incompressible Viscous Flow Problems," *Journal of Computational Physics*, Vol. 2, pp. 12–26, 1967.

[10] Dreyer, J. J., *Finite Volume Solutions to the Steady Incompressible Euler Equations on Unstructured Triangular Meshes*, Master's Thesis, Department of Mechan-

ical and Aerospace Engineering, Princeton University, June 1990.

[11] Rizzi, A. and Eriksson, L., "Computation of Inviscid Incompressible Flow with Rotation," *Journal of Fluid Mechanics*, Vol. 153, pp. 275–312, 1985.

[12] Belov, A., Martinelli, L., and Jameson, A., "A Novel Fully Implicit Multigrid Driven Algorithm for Unsteady Incompressible Flow Calculations," to be presented at the Second European Computational Fluid Dynamics Conference, Stuttgart, Germany, September 1994.

[13] Belov, A., Martinelli, L., and Jameson, A., "A New Implicit Algorithm with Multigrid for Unsteady Incompressible Flow Calculations," AIAA Paper 95-0049, AIAA 33rd Aerospace Sciences Meeting, Reno, Nevada, January 1995

[14] Baker, T.J., "Three Dimensional Mesh Generation by Triangulation of Arbitrary Point Sets," *AIAA 8th CFD conference*, pp. 255-270, 1987.

[15] Rogers, S.E. and Kwak, D., "Upwind Differencing Scheme for the Time-Accurate Incompressible Navier-Stokes Equations," AIAA Journal, Vol. 28, No. 2, 253-262, 1990.

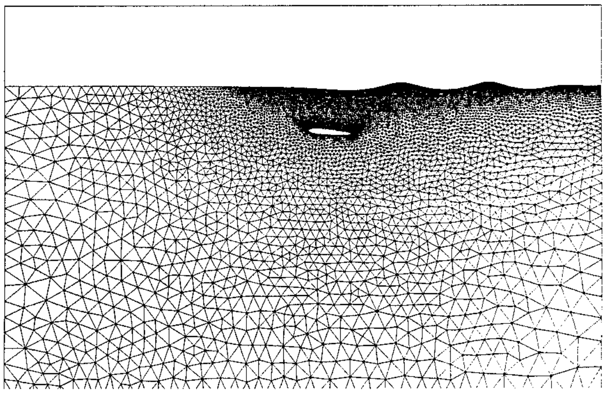
[16] Miyaka, T., Sakamoto, Y., Tokunaga, H., and Satofuka, N., "Numerical Solution of Incompressible Flow Using Two-Step, One-Stage Runge-Kutta Time Intergration Scheme ," Presented at the 1st European Computational Fluid Dynamics Conference, Brussels, Belgium, 7-11, September, 1992.

[17] Duncan, J.H., "The Breaking and Non-Breaking Wave Resistance of a Two-Dimensional Hydrofoil ," Journal of Fluid Mechanics, Vol. 126, pp. 507-520, 1983.

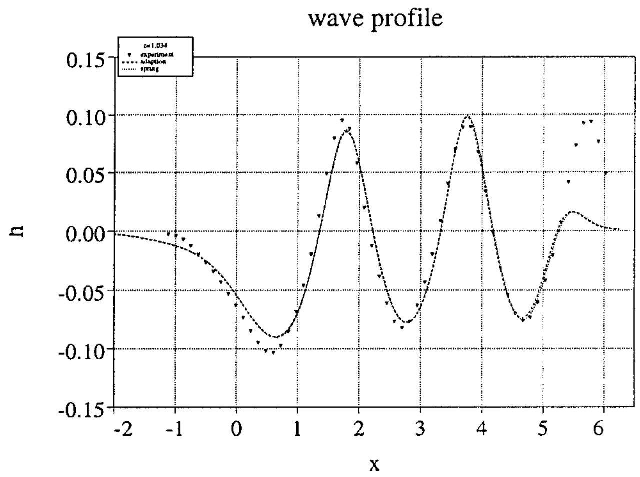
[18] Dommermuth, G.D., Yue, D.K.P., *et al.*, "Deep-Water Plunging Breakers: A Comparison Between Potential Theory and Experiments ," Journal of Fluid Mechanics, Vol. 189, pp. 423-442, 1988.

[19] Haussling, H.J. and Van Eseltine, R.T., "Finite-Difference Methods for Transient Potential Flow with Free Surface ," 1st International Conference on Numerical Ship Hydrodynamics , pp.295-313, 1975.

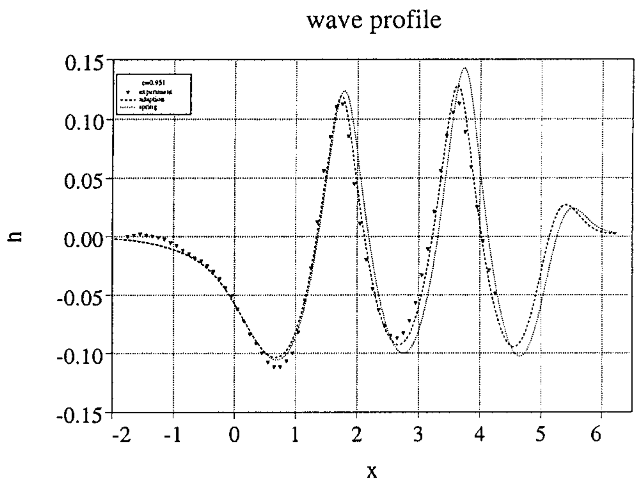
[20] Israeli, M. and Orszag, S.A., "Approximation of Radiation Boundary Conditions," Journal of Computational Physics, Vol. 41, pp. 115-135, 1981.



1a: Computationl Grid

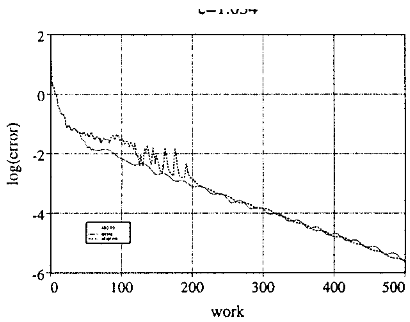


1b: Computed Wave Elevation, F = 0.567 c=1.034

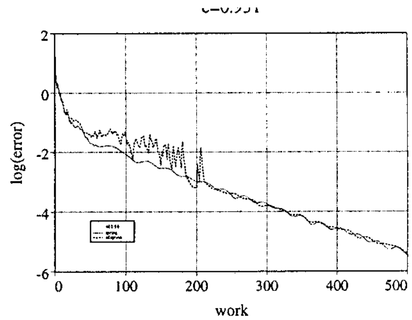


1c: Computed Wave Elevation, F = 0.567 c=0.951

Figure 1: Steady State Calculation of Wave Pattern over a NACA0012 Airfoil . Dot Line Spring Method, Dash Line Adaption Method.

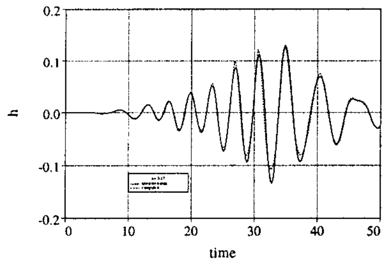


2a: Convergence History for c=1.034

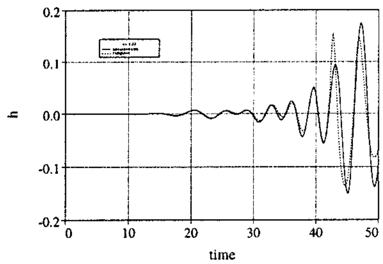


2b: Convergence History for c=0.951

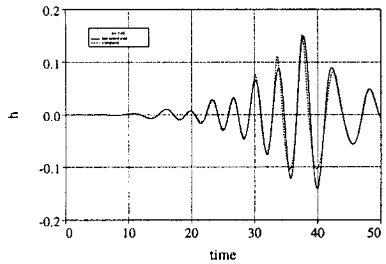
Figure 2: Convergence History. Dot Line Spring Method, Dash Line Adaption Method.



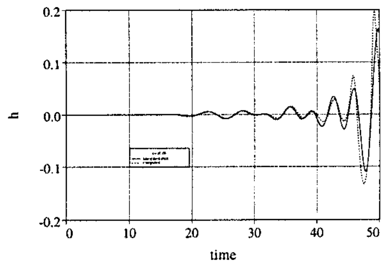
3a: station x=3.17



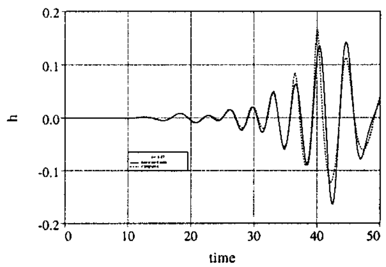
3b: station x=8.33



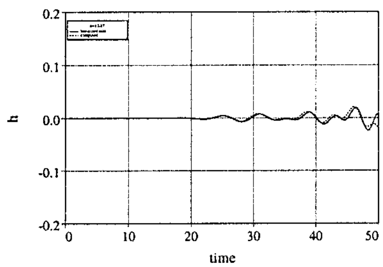
3c: station x=5.00



3d: station x=10.00



3e: station x=6.67



3f: station x=12.17

Figure 3: Wave Elevation vs Time. Dashed line Non-Linear calculation, Solid Line Linear Theory.