SPECIAL ISSUES IN UNSTEADY NAVIER-STOKES SIMULATIONS WITH DEFORMING BODIES

Jaakko Hoffren

Laboratory of Aerodynamics
Helsinki University of Technology
FIN–02150 Espoo, Finland

Abstract

The development and testing of a multi-block Navier-Stokes solver for dynamic solid/fluid interactions is reviewed. During the description of the code, special features related to the time-dependent computations are emphasized, and important issues to be considered in the development of time-accurate simulation methods are discussed. The discussion is supported by selected results from test cases, including aeroelastical problems. The purpose of the paper is to give general knowledge about the performance of unsteady flow simulations.

Introduction

Navier-Stokes solvers for steady flows are nowadays well-established and applicable to practical problems. However, a large number of important flow problems is characterized by time-dependent behaviour, requiring time-accurate simulations. Additional complications in the unsteady cases are often caused by flexible computing domains around deforming bodies. Furthermore, if the deformation of the geometry depends on the flow, a method capable of modelling solid/fluid interactions is required to simulate the case.

Time-accurate Navier-Stokes simulations have been performed since the early stages of computational fluid dynamics, but only during the past few years has the development of computers enabled rapid progress in the field. The simulation of the dynamic interaction between fluid and solid is also becoming feasible. However, due to the relatively short history of unsteady Navier-Stokes calculations, no general concensus on the practical applicability of the various possible concepts has been found, although the basic principles of the dynamic simulations are clear.

Since 1990, a time-accurate Navier-Stokes solver for compressible flows has been developed in the Laboratory of Aerodynamics at the Helsinki University of Technology. The finite-volume code working with multi-block structured grids is based on an established steady-state flow solver, known as FINFLO, which is documented, for example, in Refs. 1 and 2. The time-accurate version of the code relies on an implicit temporal discretization and an iterative time-stepping.[3,4] The flow solver can deal with arbitrarily deforming computing domains, and recently, it was coupled with interactive calculations of structural deformations.[5,6]

In this paper, the development and testing of the Navier-Stokes code for dynamic solid/fluid interactions is reviewed. During the description of the code, issues specific to the development of time-accurate simulation methods are emphasized. The available alternatives and the choices made for each feature of the present solver are discussed, and the discussion is supported by selected illustrative test results. The purpose is to disseminate general information about the essence of the unsteady flow simulations, which will hopefully guide future work in the field.

Basic Flow Solver

General Formulation

The Navier-Stokes computations are based on the fundamental conservation laws for mass, momentum and energy. Therefore, a conservative form of the numerical simulation scheme is preferable. This requirement leads naturally to a finite-volume formulation of the problem, where the physical laws are written directly for each deformable, discrete computational cell bounded by m flat segments as follows:

$$\frac{d}{dt}VU + \sum_{k=1}^{m} \vec{F}_k(U_k) \cdot \vec{S}_k = 0 \qquad (1)$$

Here V means the cell volume and $\vec{S}_k$'s are the vectorial cell surface segment areas. The column matrix U contains the conservative variables: density, momentum components and total energy. The flux vectors $\vec{F}(U) \cdot \vec{S}$ related to each cell face represent all the convection, diffusion and pressure gradient terms of the

Navier-Stokes equations, not written explicitely here for brevity. To take the cell face movements into account, the convection terms contain the cell face velocity normal components.[4]

An established solution strategy considered here, too, is to separate the spatial and temporal discretizations. The technique, known as the method of lines, leaves a wide variety of alternatives available for each subtask. The first major choice concerns the type of computational grid to be utilized. The most established choice is a non-overlapping structured multi-block grid that facilitates the use of efficient solution methods while maintaining the conservation property. On the other hand, the generation of patched grids for complex geometries is difficult and very large deformations or geometry disintegrations, like store separations, cannot be handled. The latter limitation can be avoided and the grid generation simplified by using overlapping structured grids, known as Chimera-technique. However, the flow solver becomes complicated, and the conservation property at block interfaces is difficult to maintain. A totally different approach is to employ unstructured grids that are easy to generate and adapt for complex deforming domains. Their use for high-Reynolds-number applications is, however, not straightforward and such schemes are still in an early stage of development, even for steady flows. The flow solvers based on unstructured grids also tend to be computationally relatively inefficient, which is a drawback in intrinsically heavy time-accurate simulations. The ranking of the alternative approaches depends on the applications in mind. If relatively standard, reasonably deforming bodies are to be studied at high Reynolds numbers, the structured multi-block scheme may be the most suitable. The capabilities of this type of solver can be later extended by adding the Chimera-technique. On the other hand, if very complex geometries differing widely from each other are to be studied and the Reynolds numbers are low or the viscous effects can be neglected, unstructured grids are at their best. The present flow solver is based on the first alternative, biasing the subsequent discussion accordingly.

Spatial Discretization

The cell face velocity components must be taken into account in the formulation of the spatial discretization, i.e. in the calculation of fluxes $\vec{F}(U) \cdot \vec{S}$. The addition of grid dynamics is easy in some basic flux evaluation methods, but quite complex in others.

In central-difference-type spatial discretizations, the addition of grid velocities into the convection terms is trivial. In some situations, however, the necessary artificial dissipation terms may require adjustments from their basic forms because the grid movement modifies the magnitude of relative convection. For upwind-type schemes, the complications resulting from the cell face velocities vary between flux formulations. The well-known Van Leer's scheme becomes considerably more complicated when the grid velocities are implemented into it.[7] This decreases the computational efficiency that is more important in time-accurate simulations than in steady-state cases. The overall efficiency issue may also put some other sophisticated schemes in an unfavourable position. In contrast to Van Leer's scheme, the modification for deforming grids is very easy for Roe's method, where the convection speeds are used as distinct, clearly understandable components of the flux formula. The only modification needed is a subtraction of the cell face normal velocity components from the normal flow velocities in the flux difference terms. Because of its general success in viscous calculations and suitability to deforming grids, Roe's scheme was adopted for the present flow solver.[3,8]

During the code development, test runs were performed with Van Leer's and Roe's method in a fixed grid. It turned out that Roe's scheme needed an entropy fix to prevent a possible emergence of non-physical expansion shocks. For this, the fix of Yee was successfully implemented.[3,8] The comparisons also revealed that at high supersonic Mach numbers, Van Leer's scheme worked better than Roe's, as shown in Fig. 1. However, Van Leer's scheme was not studied further.

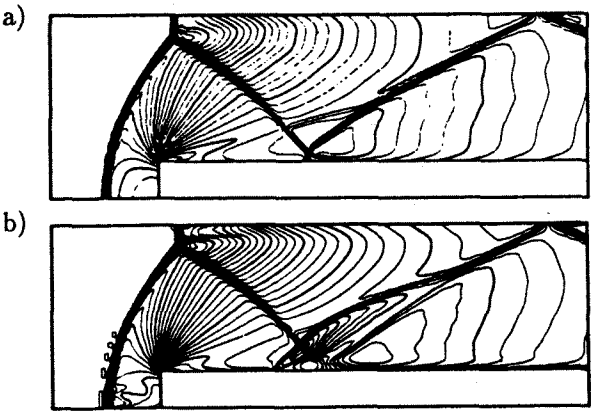

FIGURE 1 - Instantaneous density contours in a shock channel at an inflow Mach number of 3.0 computed inviscidly a) with Van Leer's scheme b) with Roe's scheme.

Reasonable simulations require at least second-order accurate spatial discretizations. In the scheme under review, the evaluation of the flow states at cell faces needed for the fluxes are computed from a MUSCL-type difference formula with an optional flux limiter.[1] The discretization is thus formally second- or third-order accurate in smooth regions of the flowfield.

The viscous and heat fluxes do not require any special consideration in dynamic simulations since their basic forms are not changed by the grid deformations. The thin-layer approximation with conventional central differences can be applied, as done in FINFLO. The tur-

bulence effects related to the viscous stresses will be discussed later.

Boundary Conditions

In external flow calculations, free-stream conditions can be specified at the grid outer boundary provided that the grid is large enough. This simple approach is chosen in the solver under study. A simple analytical circulation correction that may significantly improve the accuracy of steady-state simulations cannot take time-dependent effects into account.

At solid viscous walls, the flow velocities are to be set equal to the grid velocity, and a flow tangency condition can be applied at possible inviscid walls and symmetry planes. It is important to maintain the order of accuracy of the basic spatial discretizations, which can be done by extrapolating the wall pressure and viscosity from the flowfield with second-order accuracy and by applying non-symmetrical differences for the diffusive gradients, as done in FINFLO. The wall fluxes are then computed from their basic, non-upwinded forms.

To be able to study complex flow regions with structured grids, a multi-block strategy must be used. In FINFLO, the blocks are non-overlapping with continuous cell distributions across the interfaces to facilitate complete continuity and conservation of the solution. For the calculation of the fluxes at common boundaries of the blocks, states in the two cell rows adjacent to the interface are transferred by utilizing two layers of ghost cells outside each block overlapping the neighbouring block. Maintaining the conservation properties with dynamic Chimera-grids may be difficult. Solutions to this problem have been presented[9], but the required technique appears to be somewhat complicated and computationally intensive.

Temporal Discretization

Theoretical Considerations

Time-accurate simulations require constant time steps everywhere in the computing domain unless some special measures are taken. Since the smallest cells tend to determine the time step lengths, the computations with reasonable grids are bound to be heavy. For the temporal discretization and the related time integration of the flow equations, a wide variety of schemes has been applied. The methods can be broadly classified into explicit and implicit schemes with different strengths and weaknesses.

Owing to their relative simplicity, explicit time-integration methods have been popular from the early days of numerical flow simulations. Of these, different versions of the Runge-Kutta scheme have become best established. They are second-order accurate in time, easy to code and straightforward to vectorize, but they suffer from serious time-step stability limitations, as do all the explicit schemes. This deficiency makes them poorly suited to simulations of viscous flow at high Reynolds numbers, which require very dense grids.

The stability limitations of the time step can be avoided by using implicit schemes. When the inevitable computational complications are overcome, the time step lengths will be governed by accuracy requirements. By far the most popular implicit time-integration method has been the first-order accurate backward-Euler scheme, usually combined with an approximate factorization to facilitate the solution at each time step. The method is robust, relatively simple and appears to be practical for slowly varying situations. Calculations with second-order accurate implicit schemes are also found in the literature. The three-level fully implicit scheme has been succesfull, but the trapezoidal rule or the Crank-Nicolson scheme has suffered from stability-related problems.

In order to be able to simulate practical high-Reynolds-number viscous flows with reasonable calculation times, an implicit formulation is necessary. In the smallest boundary layer cells, the local Courant numbers may then be of the order of 10^3 without losing the global temporal accuracy. A higher-than-first-order discretization combined with an iterative solution method appears to be a promising concept for an efficient scheme. Based on these considerations, the following family of two- or three-level temporal discretizations was implemented in FINFLO:

$$(1+\gamma)(VU)^{n+1} - (1+2\gamma)(VU)^n + \gamma(VU)^{n-1}$$

$$= \Delta t[(1-\beta)R^n + \beta R^{n+1}] \qquad (2)$$

Here an abbreviation $R = -\sum_{k=1}^{m} \vec{F}_k(U_k) \cdot \vec{S}_k$ is adopted for the sum of the spatial terms, and the superscripts relate to the time levels involved. The formula with two parameters facilitates the testing of different schemes. The parameter γ controls the levels to be employed, and β defines the extent of the implicitness. By selecting certain combinations of γ and β, for example the Crank-Nicolson scheme or the three-level fully implicit scheme (3-LFI) can be specified. The drawback of this formulation is the relatively large computer memory requirement, since the solution at three time levels and the residuals R at two time levels must be storable. However, the solution for the higher-order discretizations is no more complex than for the Euler scheme.

Test Results

In initial inviscid test calculations with FINFLO, the Crank-Nicolson scheme was generally applied. In the channel case illustrated in Fig. 1, it gave similar results at a reduced effort in comparison with the explicit Euler scheme, which was still usable.[3]

Further tests were performed with more realistic, viscous and turbulent flow conditions. A suitable situation is the transonic buffet on a biconvex airfoil at $Re = 11 \times 10^6$, for which experimental results are available for comparisons.[10] In this case, a strong oscillatory interaction takes place between the boundary layer and shock waves on the fixed airfoil in a certain Mach number range, posing a challenge for numerical flow simulations. The calculations with the algebraic Baldwin-Lomax turbulence model and a 192×64-cell C-grid were successful,[4] giving results that agreed better with the experimental data than the published reference results obtained with coarser grids.

In this context, different temporal discretizations were evaluated among other computational parameters by repeating a calculation of a certain period of oscillation.[4] Only implicit discretizations were tested since at the studied Reynolds number, explicit schemes are completely impractical. The Crank-Nicolson scheme that was used in the early studies turned out to have stability problems and eventually failed. Instead, the three-level fully implicit scheme proved to offer a good combination of accuracy and robustness, being much more efficient than the first-order implicit Euler scheme. This result, illustrated in Fig. 2, was so evident that all the unsteady calculations performed with FINFLO since these tests have applied the 3-LFI scheme. No stability problems or other difficulties have ever been encountered.

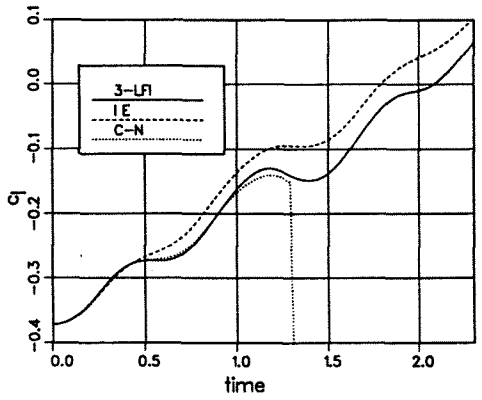

FIGURE 2 - Behaviour of lift of a biconvex airfoil at $Ma = 0.783$ during the test period computed with constant time steps and different temporal discretizations. The 3-LFI result is verified to be numerically very accurate.[4] IE refers to the implicit Euler scheme, and C-N means the Crank-Nicolson scheme that failed in the test.

Solution Method

Theoretical Discussion

Implicit temporal discretizations lead to large equation systems to be solved at each time step. Usually, the solution is based on the linearization of the residuals R^{n+1} to obtain a workable formulation. However, the linearization leads to numerical errors with non-linear flow equations, degrading the basic discretization accuracy if applied in a straightforward manner. A direct solution of the equation system is presently not feasible in realistic multi-dimensional simulations, but with structured grids an approximate factorization can be performed to obtain a solution of second-order numerical accuracy. If the temporal accuracy of the basic discretization is of second or higher order, the approximate factorization introduces significant additional errors to the solution. A proper implicit treatment of boundary conditions causes difficulties, too. Especially problematic are the block interfaces in multi-block grids, although a method facilitating continuous one-dimensional sweeps throughout the computing domain was recently published.[11] The errors discussed are specific to unsteady simulations, not affecting the accuracy of steady-state simulations with a time-integration approach.

The linearization and factorization errors can be completely eliminated, thus allowing enlarged physical time steps while retaining the accuracy of the basic temporal discretization by employing iterative solution methods. The treatment of boundary conditions and block interfaces becomes simple, too. With unstructured grids, iterations are in any case necessary to enable a solution. Several different iteration schemes have been succesfully applied, including pseudo-time integrations within the physical time steps, Gauss-Seidel variants and the GMRES method, possibly with pre-conditioning.

If a pseudo-time integration approach is chosen for the iteration, only explicit schemes, like Runge-Kutta, are applicable to unstructured grids. These can be applied with structured grids, too, but high Reynolds numbers may prove problematic. Implicit schemes applicable only in structured grids are more robust, and such a method is applied in FINFLO. The scheme is derived from Eq.(2) by linearizing it at an intermediate state k. When the new solution is written in a delta-form and the residual R^{n+1} is linearized, an equation for the corrections δU at each iteration is obtained. The equation applicable to each cell can be reduced to the form

$$(1+\gamma-\beta\frac{\Delta t}{V^{n+1}}[\frac{\partial R}{\partial U}]^k)\delta U$$

$$= \frac{1}{V^{n+1}}\{-(1+\gamma)V^{n+1}U^k+(1+2\gamma)(VU)^n-\gamma(VU)^{n-1}$$

$$+\Delta t[(1-\beta)R^n+\beta R^k]\} \qquad (3)$$

From this equation, U^{n+1} can be iterated by computing successive corrections for U^k, applying a matrix solver required by the left-hand side until the process is sufficiently converged. However, because of the approximations made in the adopted solver, Eq.(3) is modified to ensure stability of the iteration at arbitrary physical time steps. This can be accomplished without affecting the solution accuracy, because at the converged

situation δU is zero. The stabilization is made by replacing γ on the diagonal of the left-hand side with a variable weight $\Delta t/\Delta\tau$, where $\Delta\tau$ is a stability-limited local pseudo-time step related to the solver employed.

After the stabilizing modification and a subsequent multiplication by $\Delta\tau/(\Delta t + \Delta\tau)$, Eq.(3) can be converted to

$$(1 - \beta\frac{\Delta\tau_{mod}}{V^{n+1}}[\frac{\partial R}{\partial U}]^k)\delta U = \frac{\Delta\tau_{mod}}{V^{n+1}}R^k_{mod} \qquad (4)$$

where $\Delta\tau_{mod} = \Delta t\Delta\tau/(\Delta t + \Delta\tau)$ and the modified residual R^k_{mod} is the braced expression of the right-hand side of Eq.(3) divided by Δt.

Eq.(4) is of the form used in typical implicit steady-state flow solvers employing pseudo-time integration. The present solver utilizes the steady-state FINFLO[1,2] with minor modifications to obtain the corrections δU. The scheme is based on an approximate factorization and local pseudo-time steps determined by a user-defined Courant number and a diffusion stability criterion. In addition, the convergence can be accelerated by a multigrid algorithm based on the work of Jameson and Yoon, the implementation of which in FINFLO is described in detail, for example, in Ref. 1.

During the solution process, the blocks are handled sequentially within each pseudo-time step (iteration cycle), keeping all the boundary conditions fixed. After all the blocks have been computed, the block boundaries are updated. The final result is completely continuous when the calculation is sufficiently converged. As a result, the grid block interfaces and boundary conditions cause no deterioration in the basic spatial or temporal accuracy. Two convergence criteria in addtion to a specification of a fixed number of iterations were devised. In the first criterion, a limit for the maximum density correction is defined. The second one compares the L_2-norm of the density corrections at each pseudo-time step to the corresponding norm at the first step.

Test Results and Conclusions

Within the transonic buffet case discussed earlier, numerous tests related to the iteration at each physical time step were performed. The computational variables discussed here are the length of time steps, the number of iterations within time steps and the number of multigrid levels.

The effects of the time step lengths and the number of iterations within time steps were strongly interrelated. A certain total computational effort could be divided between the number of time steps or iterations within certain bounds to obtain similar results. In any case, some iterations were necessary to properly update the grid boundaries. Non-iterative calculations with very short time steps were inaccurate and inefficient. Later experience has suggested that generally, five iterations are sufficient.

In steady-state calculations with FINFLO, the multigrid algorithm employing normally five grid levels enhances the convergence rate significantly, the attainable speed-up factors being five to ten. Unfortunately, the benefit of the technique in time-accurate simulations seems to be much more modest. The tests revealed that the use of only two grid levels appears to be beneficial, the additional levels just consuming computational effort with a negligible effect on convergence. With two levels, the test computations took just about 15 per cent less effort than single-grid calculations of similar accuracy, as illustrated in Fig. 3.

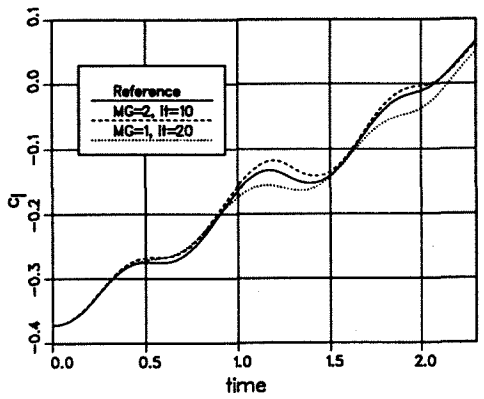

FIGURE 3 - Behaviour of lift of a biconvex airfoil at $Ma = 0.783$ during the test period computed with constant time steps and variable numbers of iterations and multigrid levels.

It can be thought that the observed poor performance of the multigrid algorithm is related to the fact that in unsteady calculations no disturbance waves are to be rapidly spread through the computing domain within physical time steps. This fundamental idea based on the requirement of temporal accuracy is substantiated by the lack of published, successful time-accurate multigrid calculations. The author is unaware of any references convincingly showing any significant benefit from the technique in this context, although a few papers on the subject have been published recently.[12-15] The issue appears to be open to debate and warrants further research.

In addition to the disappointing multigrid efficiency, the convergence criteria within physical time steps proved to be problematic. A suitable limit for the maximum density correction is case-dependent because in contrast to steady-state calculations, very low values ensuring always proper convergence cannot be applied due to resulting excessive run times. Thus, the limit value tends to depend strongly on the characteristic Mach number and geometrical severity of the problem. The relative L_2-norm of the density corrections is unreliable, too, being dependent on the instantaneous flow state. If the situation is temporarily almost steady, the

relative norm may never reach the specified limit, and during rapid changes the absolute convergence may be insufficient although the relative norm drops rapidly. It seems to be very difficult to apply a convergence criterion that would ensure sufficient convergence in different flow cases without unnecessary calculations. Therefore, FINFLO is normally run by specifying a fixed number of pseudo-time steps within each physical time step, which is certainly not optimal. The development of universal, probably hybrid convergence criteria for iterative time-accurate solvers is a topic requiring further work.

Overall, the computational efficiency of time-accurate FINFLO is not as good as was hoped for. In typical simulations involving periodic flow, 200-800 time steps including 5 iterations per step are required to achieve good temporal accuracy. It is felt that within the present framework the iteration efficiency could probably be improved, perhaps by completely chancing the principle, and the convergence criteria could certainly be developed via laborous, systematic tests.

Treatment of Grid Deformations

Although the conservation form of the flow equations is generally preferable in numerical simulations, the geometrical conservation law has often been neglected. This relationship connects the grid cell face velocities and volume changes in such a way that the grid deformations do not introduce any additional numerical errors to the solution. The importance of respecting the geometrical conservation law in practical calculations is somewhat controversial, and conflicting arguments can be found in the literature.

In developing the time-accurate FINFLO, the exact fulfilment of the conservation of geometry was enforced. A suitable mathematical formulation for the geometrical conservation law is obtained from Eq.(2) by requiring exact mass conservation in a free stream. By taking account of the specified constant density and flow velocity, an equation connecting the cell volume changes and the cell face velocities is obtained. From this basic formulation, a useful condition applicable to each cell face separately can be deduced by understanding the volume changes to mean the volumes $\Delta\mathcal{V}$ swept by the face within the time steps involved. With this reasoning, the geometrical conservation law in its final form is written as

$$(1+\gamma)\Delta\mathcal{V}^{n+1} - \gamma\Delta\mathcal{V}^{n} = \Delta t[(1-\beta)v_n^n S^n + \beta v_n^{n+1} S^{n+1}] \tag{5}$$

from which the cell face normal velocity components v_n^{n+1} can be solved after the grid node coordinates are updated.

In addition to the cell face normal velocity, the cell face tangential velocity component is needed for the calculation of viscous fluxes at the solid walls. The necessary extra condition follows from the average direction in which the cell face center is moving. Applying this idea, a unique grid velocity vector fulfilling the geometrical conservation law can be computed.

During the testing of the present solver, a direct comparison between the conservative and a simplified non-conservative treatment of the grid deformations was performed. The test case was a transonic aileron buzz, where a control surface of a NACA 6-series airfoil oscillates at a large amplitude driven by the dynamic interaction of moving shock waves and separating boundary layers.[16] As can be seen in Fig. 4, at least in this case, the geometrical conservation law and the grid velocity evaluation method were not critical factors concerning the predicted motion.[5] The practically identical results obtained with three different schemes for the cell face velocities support the claims about the minor importance of the law. On the other hand, the speed-up of the calculations resulting from the simplifications is negligible. To avoid unnecessary sources of errors in calculations involving general dynamic grids, it is still advisable to apply the easily realizable conservative scheme.

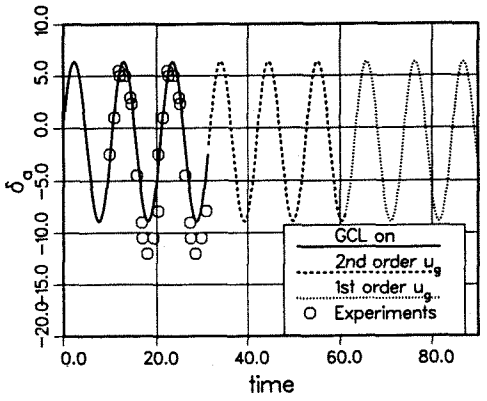

FIGURE 4 - Aileron deflection histories in a transonic aileron buzz computed with different grid velocity evaluation methods.

Turbulence Modelling

Principles of Modelling

The simulation of turbulence effects is even more problematic in unsteady flow calculations than in steady-state cases, at least until inherently time-dependent large eddy simulation (LES) becomes practical. Realistic present-day and near-future studies must be based on a time-averaging of turbulence, leading to the Reynolds-averaged Navier-Stokes equations. The time-averaging itself may be a source of trouble in some applications if the time scales of turbulence and the oscillations of the main flow interfere. If the turbulence cannot be reasonably filtered from the overall vortical flow, the models based on this assumption may behave unpredictably. Very little is known about the severity

of this potential problem, and it is normally not considered.

Evident difficulties in the turbulence modelling in time-accurate calculations are related to the computational efficiency. Very sophisticated, complex models tend to make the code impractically heavy to use. The complex models also tend not to be very robust. It is unacceptable if the user has to monitor a long computer run closely and possibly change some computational parameters during it. On the other hand, a fundamental difficulty in the time-dependent cases comes from the need to model turbulence history. Still more complications follow from possibly moving transition points. Even in steady-state applications, the modelling of transition is not a well-established technique.

The most advanced turbulence modelling methods based on the Reynolds-averaging are known as Reynolds stress models (RSM) or second-moment closures. In RSM, the apparent turbulent stresses emerging from the time-averaging are solved from individual partial differential equations for each stress component, enabling the modelling of turbulence unisotropy. In some applications, the property is shown to be crucial. However, this type of model appears to be immature for practical time-accurate simulations since little if any such results have been published. Suitable formulations even for steady-state calculations are still under debate. In any case, the models will require substantial computational resources.

A simpler approach is to apply the Boussinesq eddy-viscosity approximation to the turbulent stresses. The approximation reduces the modelling to a modification of the effective viscosity in the viscous terms of the laminar-like equations. An inevitable side-effect is the assumption of isotropic turbulence, which in some cases, especially in those involving swirling flows, spoils the modelling realism.

The most sophisticated models based on the Boussinesq-approximation rely on two partial differential equations for selected turbulence-related quantities, one of which is normally the turbulence kinetic energy. These quantities together define the addition to the effective viscosity. The time-dependent formulation of the two-equation turbulence models gives them a capability to take the history effects into account, which makes them physically reasonable in time-accurate simulations. In addition, two-equation models are not in principle tied to the concept of boundary layer, and they are relatively straightforward to apply in multi-block or unstructured grids. However, the models can be computationally relatively inefficient, they may have difficulties in the near-wall treatment, and mixed laminar/turbulent flows cause practical complications in their application. In particular, moving transitions are difficult to handle with them.

Because the second equation besides the one for the turbulence kinetic energy in all two-equation models is somewhat heuristic, it can be argued that a model based on just one partial differential equation could work about as well. This is the motivation of one-equation turbulence models, in which there is currently considerable interest. New formulations designed specifically for aeronautical Navier-Stokes simulations involving partially laminar flow, like the Spalart-Allmaras model,[17] have recently emerged. This type of model, having basically all the capabilities of the two-equation models while being computationally more efficient and robust, appears quite promising. However, there is not yet much relevant experience to draw firm conclusions.

The simplest methods of simulating the turbulence effects are the algebraic turbulence models. The turbulent viscosity predicted by them depends only on the local and instantaneous flow conditions. In steady-state cases, the limited dependency is often justified, but the total lack of history effects in turbulence modelling makes the algebraic formulations at least doubtfull in time-dependent studies. General deficiencies of these models are the incapability to deal with separated flows and practical application difficulties in multi-block or unstructured grids. On the positive side, algebraic turbulence models are robust and computationally efficient, transition can be easily defined, and their accuracy within their calibrated application range is known to be relatively good in steady-state calculations.

Turbulence Model Tests with FINFLO

Todate, four turbulence models have been tested in time-dependent calculations with FINFLO. The simplest of them are the two most established algebraic models in aeronautical CFD, namely the Baldwin-Lomax and the Cebeci-Smith model. Of the two-equation models utilized, Chien's low-Reynolds-number $k-\epsilon$ model[18] is well-known. It is also widely accepted that the model is not particularly suitable to aerodynamics with characteristically strong pressure gradients and flow separations. A new two-equation model giving promising results for steady aeronautical flows, even when flow separation is involved, is the $k-\omega/\epsilon$ model developed by Menter.[19] The model switches between an ω equation and an ϵ equation dynamically in an effort to combine their best features. The switching appears to make the formulation rather delicate for unsteady flows. In addition, the eddy viscosity limitation of the model, called SST (Shear Stress Transport), tends to further reduce its robustness. Nevertheless some calculations were performed with Menter's model for comparisons with Chien's model.

Several test cases were studied with the algebraic models. The Baldwin-Lomax model was applied to the

transonic buffet discussed earlier and to airfoils oscillating in pitch in transonic conditions.[20] The computations themselves were straightforward, and the results obtained agree well with available, corresponding reference calculations, but only fairly well with experimental results.[4] However, the observed discrepancies cannot be said to be only related to turbulence modelling since there are some obvious anomalies in the experimental data.[20] Because no further calculations with more advanced models were performed with FINFLO or found in the literature, it is not known whether an improvement in the turbulence modelling would reduce the differences between simulations and experiments.

The Cebeci-Smith model was applied to two aeroelastical problems. The first case is the transonic aileron buzz[16] related to Fig. 4, and the second one involves a two-degree-of-freedom flutter of a NACA 64A010 airfoil.[21] In both cases, trouble-free computations with 192 × 64-cell grids gave similar results to those of the reference calculations.[16,21]

The aileron buzz case was also studied with Chien's and Menter's models.[6] It turned out that the simulations utilizing the two-equation models consumed about a six-fold computational effort compared to the Cebeci-Smith model for similar temporal accuracy. The two-equation models required about four times as many iterations per time step as the algebraic model to obtain sufficient convergence within constant time steps, and each iteration with two extra equations consumes about 50 % more computing time than the operation with the mean flow equations only.

The regular aileron oscillation amplitudes obtained with Chien's and Menter's models differed somewhat from the Cebeci-Smith results at the nominal Mach number of 0.82. The result obtained with Chien's model agrees best with the measurements by reaching the largest negative deflections, as seen in Fig. 5a. Much larger differences between the models are seen in the aileron hinge moment histories of Fig. 5b, where especially the curve obtained with Menter's model looks curious. In careful, additional studies it was verified that the result is really temporally quite accurate. Thus, it is concluded that the observed behaviour is characterictic of Menter's model as implemented in FINFLO. A detailed study of the flowfield revealed that the oscillations in the hinge moment are related to a shedding of pressure disturbances from the shock root into the shear layer, which in turn is probably caused by a periodic switching between the ω and ϵ equations in certain local flow conditions. Although reliable reference data is lacking, it appears that Menter's model is not well-suited to flow cases of the type studied, and it was not applied in further tests.

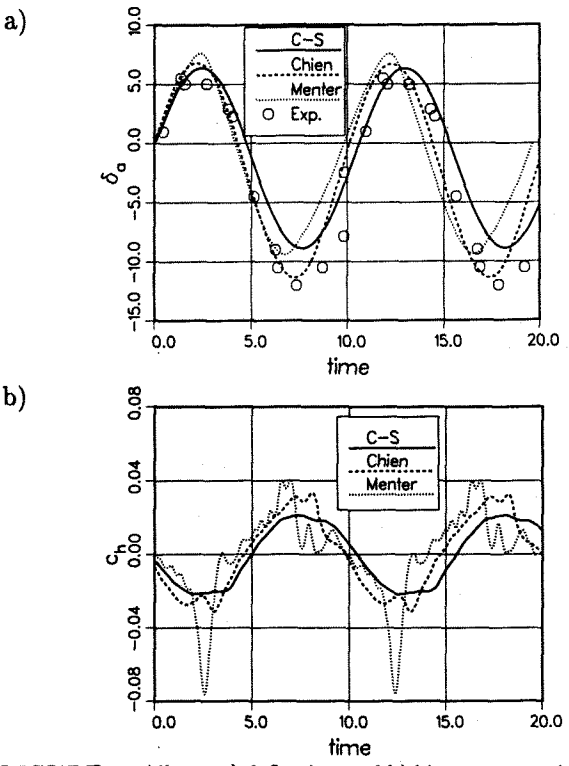

FIGURE 5 - Aileron a) deflection and b) hinge moment histories in a transonic aileron buzz computed with different turbulence models.

At $Ma = 0.82$, the oscillations with Chien's model started from the zero aileron deflection, as observed in the wind tunnel, whereas the Cebeci-Smith model predicted a decaying motion with this initial condition. Extra calculations were performed at varying Mach numbers to check the difference in the motion stability predicted by the two turbulence models. At $Ma = 0.83$, the algebraic model produced a weakly growing oscillation initiated from the steady state, and at $Ma = 0.81$, the oscillation was strongly damped. However, at this lower Mach number, Chien's model still gave a diverging motion. Thus, the stability boundaries predicted by the models differ significantly from each other, which is an unpleasant outcome.

In the flutter case with small oscillation amplitudes and no flow separation, the turbulence model should not have any effect on the solution. As expected, the computations with the Cebeci-Smith model and Chien's model gave practically identical results. However, the computations with the two-equation model required again a six-fold computational effort compared to the ones with the algebraic model to achieve similar, sufficient accuracy.[6]

Structural Coupling

Temporal Discretization

To calculate structural deformations or rigid body

movements, the solid-state equation of motion must be applied to each structural degree of freedom. In a general case, this results in a coupled matrix equation. By omitting the matrix notation, the equation can be expressed in the simple conceptual form

$$m\ddot{q} + c\dot{q} + kq = A \tag{6}$$

where m denotes a mass quantity related to each degree of freedom q, c refers to structural damping, and k is the stiffness or spring constant. The structural properties c and k may be non-linear functions of the motion driven by the flow-induced force represented by A.

For the temporal discretization and time integration of the equation of motion, a method of comparable accuracy to the accompanying flow simulation should be applied to obtain a balanced, efficient system. Computations with several explicit and implicit formulations of varying complexity have been published,[21-24] but the benefits of added sophistication have not been demonstrated in realistic simulations. In any case, simplicity and an easy coupling between the flow analysis and the structural analysis are desirable properties.

Based on the above considerations, a simple explicit second-order accurate time-integration scheme for the structure was adopted for the solid/fluid interaction tests with FINFLO. The acceleration and velocity are differenced centrally around time level n and substituted into Eq.(6). Utilizing the aerodynamic forces known at the time level n, too, the structural coordinates q at the new time level $n+1$ can be readily solved. In a one-degree-of-freedom case or with uncoupled structural modes, the solution can be written directly as

$$q^{n+1} = \frac{(4m - 2\Delta t^2 k)q^n - (2m - c\Delta t)q^{n-1} + 2\Delta t^2 A^n}{(2m + c\Delta t)} \tag{7}$$

where the superscripts refer to the time levels. When the new structural coordinates have been solved, the grid for the flow solver can be updated and the grid velocities evaluated. Subsequently, a new flow solution at the time level $n+1$ can be computed to start a new cycle.

The accuracy of the structural time integration was studied in a case involving a lateral oscillation of a circular cylinder in a low-speed laminar flow.[6] Before the coupled solid/fluid interaction calculations, the scheme based on Eq. (7) was studied by a separate program that computed the cylinder movement as driven by a prescribed sinusoidally varying external force. For this single-degree-of-freedom case containing some viscous damping, an analytical reference solution can be found. The test runs confirmed that the numerical method is quite accurate even for large amplitude oscillations with time step lengths expected to be required by the related flow calculations, as shown in Fig. 6.

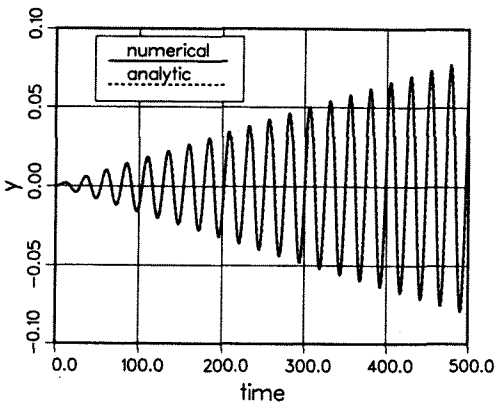

FIGURE 6 - Numerical and analytical solutions for forced, growing oscillations of a cylinder near resonance.

The subsequent actual simulations of the solid/fluid interaction produced results that agree very well with a reference[25] in every aspect. However, for accurate predictions, the time step length had to be reduced from the value that was found sufficient for the structure only. The necessary step length corresponding to about 125 steps per oscillation cycle was 2/5 of the value required by the integration of the cylinder motion.

Other solid/fluid interaction test cases simulated with FINFLO are the aileron buzz and the two-degree-of-freedom flutter discussed earlier. These turbulent cases required 200-500 steps per cycle to achieve sufficient accuracy for the flow simulation even with prescribed structural behaviour.

Based on the test results it is concluded that the flow solution clearly dominates the interaction simulations and determines their overall accuracy. The simple structural time integration scheme tested is still a stronger link than the flow solution in the complete system.

Structural Modelling

Simple structural models having just a few degrees of freedom can be connected directly to the flow solver, and the solution of the equation system derived from Eq.(6) can be written in a closed form. For more general cases involving deformations, a finite-element model is to be employed. It is reasonable to assume that the dynamic model of the structure for any practical coupled simulation can be represented by a few hundred degrees of freedom at most. Consequently, the algebraic equation system related to the updating of the structure can be easily solved by common mathematical library routines. For complex structures, a separate analysis before the coupled calculations may be necessary to reduce the size of the model.

Structural non-linearities do not pose any problems in time-integration methods where the damping and stiffness properties can be re-evaluated at each time step. However, if the structure behaves linearly, natural vibration modes can be efficiently used as the degrees of freedom. In addition to the reduced computational effort, this approach enables the use of measured structural properties, for example from a ground vibration test, instead of the calculated ones. In some cases, this option may save a large amount of work. Suitable interpolation and lumping schemes are to be utilized to convert the flow-induced forces into the form needed by the structural model. The conservation of mechanical energy must be considered in this task to maintain accuracy.[26]

Conclusions

The development of a time-accurate Navier-Stokes solver and its coupling to solid/fluid interaction has been reviewed. Related to each component of the simulation method, available options and their favourable and problematic features have been discussed, emphasizing the issues specific to time-dependent simulations.

The ranking of possible grid types is probably application-dependent, but it seems likely that in the near future structured multi-block grids will still be a reasonable option for typical aeronautical problems. This is largely due to their computational efficiency that is more important in inherently heavy time-dependent simulations than in steady-state studies. At least partially related to the efficiency is also the selection of the spatial discretization method that should allow an easy treatment of deforming grids.

Practical high-Reynolds-number simulations require an implicit temporal discretization. Test calculations with the solver reviewed demonstrated that the three-level fully implicit temporal discretization offers a good combination of robustness, accuracy and efficiency. The simple implicit Euler scheme is much less efficient, and the Crank-Nicolson scheme is prone to stability problems. An iterative solution at each physical time step appears favourable, enabling an easy and accurate treatment of boundary conditions and possible grid block interfaces. However, to devise an efficient iteration is not straightforward. The multigrid technique, highly efficient in steady-state applications, does not seem to offer any dramatic benefits in time-accurate simulations. It is also difficult to find universal and reliable convergence criteria that would ensure just enough iterations.

Moving or deforming grids are relatively easy to cope with. Although the geometrical conservation law does not appear to be a critical factor, it is recommended to be fulfilled to avoid unnecessary sources of errors since this can be done at negligible computational cost.

The most serious source of errors in the unsteady flow simulations is the modelling of turbulence, as in computational aerodynamics in general. A specific issue related to the solid/fluid interaction calculations is the predicted stability of the motion, which can depend directly on the turbulence model in use. The importance of computational efficiency related to turbulence model types is also amplified in long time-accurate calculations. The simulations with algebraic models are relatively efficient and the related results may be good, but the inability to take turbulence history effects into account degrades the reliability of such calculations. The physically more sound two-equation models have the potential to be somewhat more reliable if formulated robustly, albeit at significantly increased computational cost. One-equation models appear promising as engineering tools, too, but Reynolds-stress models will probably be restricted to special purposes in the field of time-dependent studies. In conclusion, it is felt that most leverage in further turbulence model development for unsteady flows may be found in improved versions of continuously formulated one- and two-equation models.

In solid/fluid interaction simulations, the time integration of the structural responses should be at least as accurate as the flow solution. This requirement is easy to fullfil since the flow simulation appears to be the critical element of the combination. Based on the results of three test cases studied with FINFLO, it seems that the simple second-order accurate explicit time-integration scheme chosen for the solid state equation of motion is sufficiently accurate for interaction simulation. Since the time steps for the present structural integration could be more than twice as long as the ones required by an accurate flow solution, it appears that more sophisticated schemes just consume unnecessary computational effort.

As final comments concerning the flow solver reviewed, it is considered to be applicable to a wide variety of flow conditions, basically quite accurate and very robust. The turbulence modelling is an obvious weakness but certainly not unique to FINFLO. The computational efficiency of the method is reasonable but could probably be enhanced by a fine-tuning or more fundamental changes of the iterative time-stepping, requiring further research.

References

1. Siikonen, T., Hoffren, J., Laine, S., "A Multigrid LU Factorization Scheme for the Thin-Layer Navier-Stokes Equations," *ICAS 90-6.10.3*, Stockholm, Sweden, 1990.

2. Siikonen, T., Kaurinkoski, P., Laine, S., "Transonic

Flow over a Delta Wing Using a $k - \epsilon$ Turbulence Model," *ICAS 94-2.3.2*, Anaheim, California, 1994.

3. Hoffren, J., Siikonen, T., "Time-Accurate Calculations with an Implicit Euler/Navier-Stokes Solver," *Numerical Methods for Fluid Dynamics 4*, edited by K.W. Morton and M.J. Baines, Clarendon Press, 1992, pp. 575-581.

4. Hoffren, J., Siikonen, T., Laine, S., "Conservative Multi-Block Navier-Stokes Solver for Arbitrarily Deforming Geometries," *Journal of Aircraft*, Vol. 32, No. 6, 1995, pp. 1342-1350.

5. Hoffren, J., "Dynamic Solid/Fluid Interaction Using a Navier-Stokes Method for Deforming Grids" *Numerical Methods for Fluid Dynamics 5*, edited by K.W. Morton and M.J. Baines, Clarendon Press, 1995, pp. 423-430.

6. Hoffren, J., "Application of a Navier-Stokes Method to Dynamic Solid/Fluid Interaction," *Numerical Methods in Laminar and Turbulent Flow, Volume IX*, edited by C. Taylor and P. Durbetaki, Pineridge Press, 1995, pp. 1551-1562.

7. Parpia, I.H., "van Leer Flux Vector Splitting in Moving Coordinates," *AIAA Journal*, Vol. 26, No. 1, 1988, pp. 113-115.

8. Hoffren, J., " A Numerical Method for Simulating Unsteady Flow Including Solid/Fluid Interaction," Helsinki University of Technology, Laboratory of Aerodynamics, Report No. A-16, 1995 (Doctor's Thesis).

9. Wang, Z.J., "Conservative Interface Algorithm for Overlapped Structured/Structured, Structured/Unstructured Grids," *Numerical Methods for Fluid Dynamics 5*, edited by K.W. Morton and M.J. Baines, Clarendon Press, 1995, pp. 253-262.

10. McDevitt, J.B., Levy, L.L., Deiwert, G.S., "Transonic Flow about a Thick Circular-Arc Airfoil," *AIAA Journal*, Vol. 14, No. 5, 1976, pp. 606-613.

11. Rosenfeld, M., Yassour, Y., "Fully Implicit Solution of Time-Dependent Partial Differential Equations in Arbitrary Multizone Domains," *AIAA Journal*, Vol. 33, No.5, 1995, pp. 859-863.

12. Melson, N.D., Sanetrik, M.D., Atkins, H.L., "Time-Accurate Navier-Stokes Calculations with Multigrid Acceleration," Sixth Copper Mountain Conference on Multigrid Methods, Copper Mountain, Colorado, 1993.

13. Alonso, J., Martinelli, L., Jameson, A., "Multigrid Unsteady Navier-Stokes Calculations with Aeroelastic Applications," *AIAA Paper 95-20048*, Reno, Nevada, 1995.

14. Belov, A., Martinelli, L., Jameson, A., "A New Implicit Algorithm with Multigrid for Unsteady Incompressible Flow Calculations," *AIAA Paper 95-20049*, Reno, Nevada, 1995.

15. Arnone, A., Liou, M.-S., Povinelli, L.A., "Integration of Navier-Stokes Equations Using Dual Time Stepping and a Multigrid Method," *AIAA Journal*, Vol. 33, No.6, 1995, pp. 985-990.

16. Steger, J.L., Bailey, H.E., "Calculation of Transonic Aileron Buzz," *AIAA Journal*, Vol. 18, No. 3, 1980, pp. 249-255.

17. Spalart, P.R., Allmaras, S.R., "A one-equation turbulence model for aerodynamic flows," *Le Recherche Aerospatiale*, No. 1, 1994, pp. 5-21.

18. Chien, K.Y., "Predictions of Channel and Boundary-Layer Flows with a Low-Reynolds-Number Turbulence Model", *AIAA Journal*, Vol. 20, No. 1, 1982, pp. 33-38.

19. Menter, F.R., "Zonal Two Equation $k - \omega$ Turbulence Model for Aerodynamic Flows", *AIAA Paper 93-2906*, Orlando, Florida, 1993.

20. *Compendium of Unsteady Aerodynamic Measurements*, AGARD-R-702, 1982.

21. Nakamichi, J., "Some Computations of Unsteady Navier-Stokes Flow Around Oscillating Airfoil/Wing", Technical Report of National Aerospace Laboratory TR-1004T, Japan, 1988.

22. Azevedo, J.L.F., "Three Dimensional Flow Simulation with Application to Aeroelastic Analysis," *ICAS 88-4.10.2*, Jerusalem, Israel, 1988.

23. Guruswamy, G.P., "Unsteady Aerodynamic and Aeroelastic Calculations for Wings Using Euler Equations," *AIAA Journal*, Vol. 28, No. 3, 1990, pp. 461-469.

24. Robinson, B.A., Batina, J.T., Yang, H.T.Y, "Aeroelastic Analysis of Wings Using the Euler Equations with a Deforming Mesh," *Journal of Aircraft*, Vol. 28, No. 11, 1991, pp. 781-788.

25. Mittal, S., Tezduyar, T.E., "A Finite Element Study of Incompressible Flows Past Oscillating Cylinders and Aerofoils," *International Journal for Numerical Methods in Fluids*, Vol. 15, 1992, pp. 1073-1118.

26. Guruswamy, G.P., Byun, C., "Direct Coupling of Euler Flow Equations with Plate Finite Element Structures," *AIAA Journal*, Vol. 33, No. 2, 1995, pp. 375-377.