Melbourne Graduate Fluids Conference
Monash University, Melbourne, Australia

Validation of a Code for Aeroelasticity Turbomachinery

Ivan McBean, Kerry Hourigan, Mark Thompson

Department of Mechanical Engineering
Monash University, Clayton, Victoria, AUSTRALIA

Feng Liu

Department of Mechanical and Aerospace Engineering
University of California, Irvine, CA 92697-3975 USA

ABSTRACT

A multiblock method is presented for the solution of a three dimensional model of aeroelasticity in a turbomachine blade row. The method employs a fully coupled approach and the structural model involves modal reduction. This paper presents the computational model and its validation.

1. INTRODUCTION

As designers in the turbomachinery industry strive to design machines that are lighter, more powerful and more efficient, blade flutter has become one of the most important limiting factors on the design process. The aeroelastic response is a complex phenomenon that is not well modeled or predicted by current design techniques. Codes that implement 2-dimensional models can simulate this behaviour in a meridional plane, however the flow structures found in blade passages are generally three dimensional and such models provide a qualitative rather than quantitative analysis. Furthermore, important flow phenomena are not modeled including hub and casing vortices and tip effects.

A structured 3-dimensional Navier-Stokes code is developed to solve the unsteady governing equations. These are solved using an explicit Runge-Kutta scheme, implementing residual averaging and multigrid. The problem is then solved in a time accurate manner through a fully implicit scheme as proposed by Jameson [6]. This scheme has already been used in a 2-dimensional model of aeroelasticity in turbomachinery [15, 7]. The development of the present code is an extension of the previous 2-dimensional method to 3 dimensions. Similar algorithms have been successfully implemented in a 3-dimensional Navier-Stokes external solver that models flow over a flexible wing [16, 8].

2. FLUID MODEL

The present 3 dimensional multiblock and parallel code has been developed from a proven steady solver designed to model turbomachinery cascade flow [10, 9, 13, 17, 14, 11]. The governing equations for the unsteady fluid problem in a Eulerian reference frame with a moving mesh.

$$\frac{\partial}{\partial t}\iint_{\Omega} \mathbf{w}\, d\Omega + \oint \mathbf{f}\, dS_x + \mathbf{g}\, dS_y + \mathbf{h}\, dS_z = 0 \qquad (1)$$

where

$$\mathbf{w} = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho E \end{pmatrix} \qquad (2)$$

$$\mathbf{f} = \begin{pmatrix} \rho\bar{u} \\ \rho u\bar{u} + p \\ \rho v\bar{u} \\ \rho w\bar{u} \\ \rho E\bar{u} + pu \end{pmatrix}, \quad \mathbf{g} = \begin{pmatrix} \rho\bar{v} \\ \rho u\bar{v} \\ \rho v\bar{v} + p \\ \rho w\bar{v} \\ \rho E\bar{v} + pv \end{pmatrix}, \quad \mathbf{h} = \begin{pmatrix} \rho\bar{w} \\ \rho u\bar{w} \\ \rho v\bar{w} \\ \rho w\bar{w} + p \\ \rho E\bar{w} + pw \end{pmatrix} \qquad (3)$$

The time dependent and semi-discrete form of the governing equations may be written as

$$\frac{d\mathbf{w}}{dt} + R(\mathbf{w}) = 0 \tag{4}$$

A dual time stepping scheme [6] is used to calculate the unsteady flow problem. A second order accurate, fully implicit scheme is used to integrate Equation (4) to evolve the unsteady problem in a time accurate manner.

The discrete form of (4) is

$$\frac{3w^{n+1} - 4w^{n} + w^{n-1}}{2\Delta t} + R(w^{n+1}) = 0 \tag{5}$$

This equation may be recast into

$$\frac{d\mathbf{w}}{dt^{*}} + R^{*}(\mathbf{w}) = 0 \tag{6}$$

where

$$R^{*}(\mathbf{w}) = \frac{3w}{2\Delta t} + R(w) - \frac{2}{\Delta t}w^{n} + \frac{1}{2\Delta t}w^{n-1} \tag{7}$$

The steady state solution w in equation (6) is then equivalent to the time accurate solution w^{n+1} to equation (5). Any efficient algorithm may be used to obtain the steady-state solution to (6). In this paper, the above mentioned Runge-Kutta type scheme with multigrid is used. Minimum modification of the steady solver to make it time accurate in the above manner.

3. STRUCTURAL MODEL

Modal decomposition, otherwise known as the Rayleigh Ritz approach, reduces the structural problem to a series of uncoupled, second order differential equations. These are reduced to first order differential equations and solved by the same dual time stepping method as that for the flow equations. The problem is first solved in pseudo time using a Runge-Kutta scheme, then advanced it time through an implicit time accurate formulation [1, 2].

4. MULTIBLOCK & PARALLEL IMPLEMENTATION

A multiple block method involving of structured grids is used to make the best use of computational resources and to allow the generation of grids for complex geometries. While each block consists of a structured grid, the blocks can be connected to each other in an unstructured manner provided the mesh geometry is matched at the block interfaces. The Message Passing Interface (MPI) is used for interblock communication.

5. MODEL VALIDATION

5.1 Unsteady Cylinder

The low Reynolds number, unsteady cylinder is a well documented case in both experimental and numerical fields. In this case it was used to check the time accuracy of the unsteady implementation. A single block O-grid was generated with the far-field boundary approximately 50 chords from the cylinder surface. The code may only calculate for 3-dimensional mesh geometries, so 2 cells were used in the span-wise direction. The Mach number of the compressible solver was set at 0.2 as at this value, the effects of compressibility are assumed to be negligible. The Strouhal number for a grid of 196 x 96 x 3 was calculated as 0.181, which is close to the experimental value of 0.182.

5.2 Forced Airfoil Oscillation

To demonstrate the validity of the moving mesh, multiblock and unsteady implementation, the NACA64A010 case is presented. Computational results are compared in Figure 1 for different configurations also with experimental results. An unsteady Euler calculation is performed in the flow solver. In the first case, a single block O-grid is used in combination with TFI to deform the grid to the oscillating airfoil. The far field boundary remains rigid. The second case involves a mesh that is not deformed, but rotates rigidly with the displacement of the airfoil surface. For the third calculation, the same grid as used for the single block cases is divided into 32 equal blocks, with 4 blocks in the radial direction and 8 in the circumferential direction. In this case the block corners were located using the spring analogy. The results for inviscid flow compare similarly with results presented elsewhere [12, 1] and there is little difference between the results for the different configurations.

5.3 Coupled Airfoil Oscillation

The modeling of aeroelasticity requires the simulation of the interaction of elastic member with an unsteady flow. One of the simplest examples is Isogai's wing model [4, 5], a 2-dimensional NACA64A010 airfoil that has been

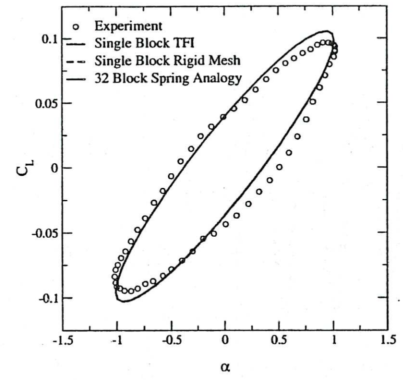
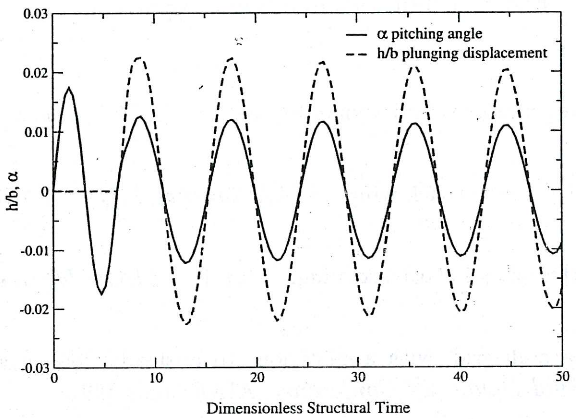


Figure 1: Comparison of unsteady NACA64A010 results with experiment.



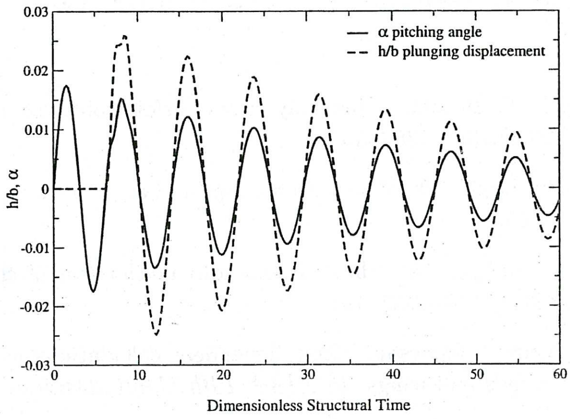


Figure 2: Stable case; $M_\infty = 0.825$, $V_f = 0.630$. Figure 3: Damped case; $M_\infty = 0.825$, $V_f = 0.530$.

studied numerically by a number of authors [1, 8]. Experimental unsteady flow measurements are available for the airfoil for forced oscillation and these results may be used to validate the unsteady flow model.

The wing is modelled by connecting a spring to the torsion and plunging axis of the NACA64 airfoil section. The section is held by a pin at the torsion axis and force for one period of oscillation whereupon the pin is released and the amplitudes of pitching and plunging. The behaviour depends on the non-dimensional parameter of flutter velocity. An example of neutrally stable, unstable and damped configurations and results are shown in Figures 2 to 4. The a plot of the flutter boundary of the configuration is shown in Figure 5, the point at which the configuration changes from being stable to unstable.

6. CONCLUSION

A novel multiblock and parallel, integrated structural and fluid solver has been presented. The implementation is general and is not limited to particular geometries and thus is flexible in that it may be applied to a broad range of problems. The moving mesh and structural model allow for the coupled solution of aeroelastic problems.

A number of different cases have been presented that compare computed results with experiment or other numerical results. Navier-Stokes solution of the flow past a circular cylinder compares well with experimental data. The code is also validated for the unsteady flow around a pitching airfoil with either a rigid grid or a deforming grid generated by a multiblock TFI method. Coupled flow and structure solution for an airfoil with two-degrees of freedom demonstrate the ability of the code to simulate coupled aeroelasticity.

References

[1] Juan Alonso and Antony Jameson. Fully-implicit time-marching aeroelastic solutions. In *AIAA 32nd Aerospace Sciences Meeting*. AIAA, January 1994. 94-0056.

[2] Juan Jose Alonso. *Parallel Computation of Unsteady and Aeroelastic Flows Using an Implicit Multigrid-Driven Algorithm*. PhD thesis, Pinceton University, June 1997.

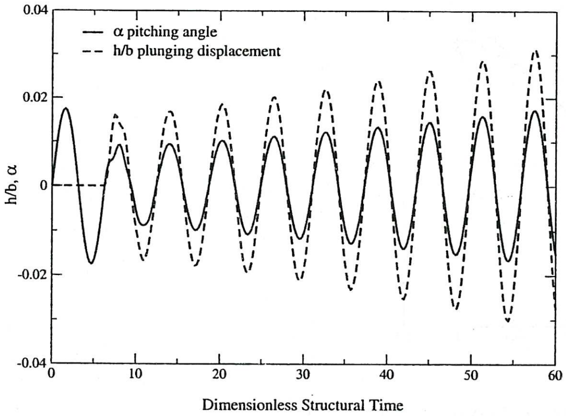


Figure 4: Diverging case; $M_\infty = 0.825$, $V_f = 0.725$.

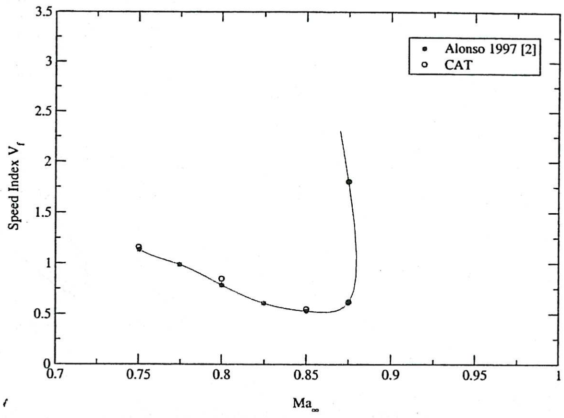


Figure 5: Flutter boundary comparison.

[3] J. T. Batina. Unsteady Euler airfoil solutions using unstructured dynamic meshes. *AIAA Journal*, 28(8):1381–1388, 1990.

[4] K. Isogai. On the transonic-dip mechanism of flutter of a sweptback wing. *AIAA Journal*, 17(7):793–795, 1979.

[5] K. Isogai. On the transonic-dip mechanism of flutter of a sweptback wing: Part ii. *AIAA Journal*, 19(7):1240–1242, 1981.

[6] Antony Jameson. Time dependent calculations using multigrid, with applications to unsteady flows past airfoils and wings. In *AIAA 10th Computational Fluid Dynamics Conference*. AIAA, June 1991.

[7] Shanhong Ji and Feng Liu. Flutter computation of turbomachinery cascades using a parallel unsteady Navier-Stokes code. *AIAA Journal*, 37(3):320–327, March 1999.

[8] F. Liu, J. Cai, Y. Zhu, and H. M. Tsai A. S. F. Wong. Calculation of wing flutter by a coupled CFD-CSD method. In *AIAA 38th Aerospace Sciences Meeting & Exhibit*. AIAA, January 2000. AIAA 2000-0907.

[9] Feng Liu and Antony Jameson. Cascade flow calculations by a multigrid Euler method. *Journal of Propulsion and Power*, 9(1):90–97, January 1993.

[10] Feng Liu and Antony Jameson. Multigrid Navier-Stokes calculations for three-dimensional cascades. *AIAA Journal*, 31(10):1785–1791, October 1993.

[11] Feng Liu, Ian K. Jennions, and Antony Jameson. Computation of turbomachinery flow by a convective-upwind-split- pressure (CUSP) scheme. In *36th Aerospace Sciences Meeting and Exhibit*. AIAA, January 1998.

[12] Feng Liu and Shanhong Ji. Unsteady flow calculations with a multigrid navier-stokes method. *AIAA Journal*, 34(10):2047–2053, October 1996.

[13] Feng Liu and Xiaoqing Zheng. Staggered finite volume scheme for solving cascade flow with a k-ω turbulence model. *AIAA Journal*, 32(8):1589–1597, August 1994.

[14] Feng Liu and Xiaoquing Zheng. A strongly coupled time-marching method for solving the Navier- Stokes and k-ω turbulence model equations with multigrid. *Journal of Computational Physics*, 128:289–300, 1996.

[15] M. Sadeghi and F. Liu. Computation of mistuning effects on cascade flutter. In *AIAA 38th Aerospace Sciences Meeting & Exhibit*. AIAA, January 2000. AIAA 2000-0230.

[16] A. S. F. Wong, H. M. Tsai, J. Cai, Y. Zhu, and F. Liu. Unsteady flow calculations with a multi-block moving mesh algorithm. In *AIAA 38th Aerospace Sciences Meeting & Exhibit*. AIAA, January 2000. To be Presented, AIAA 2000-1002.

[17] Xiaoqing Zheng and Feng Liu. Staggered finite volume scheme for solving Navier-Sokes and k-ω turbulence model equations. *AIAA Journal*, 33(6):991–998, June 1995.