Mathematical Modelling and Analysis 2005. Pages 111–116
Proceedings of the 10^{th} International Conference MMA2005&CMAM2, Trakai
ISBN 9986-05-924-0

AEROELASTIC TOOL FOR FLUTTER SIMULATION

P. POSADZY, M. MORZYŃSKI and R. ROSZAK

Division of Methods of Machine Design, Poznan University of Technology
Piotrowo 3, 60-965 Poznań, Poland
E-mail: `posadzy@stanton.ice.put.poznan.pl`

Abstract. Simulation of an aeroelastic behaviour of aerospace structures needs special approach because it combines two major disciplines: Computational Fluid Dynamics and Computational Structure Dynamics. In the present work an aeroelastic solver, i.e. arbitrary Euler/Navier-Stokes code coupled with own structural code MF3 [4] has been developed for flutter simulation. The solver has a modular structure, thus each code is run separately and then coupling process is done. The difficult problem encountered in aeroelastic computations, addressed in this paper is the coupling of solvers operating on different (CFD and CSD) meshes. Although, several tools for exchanging structural deflections to fluid mesh exist, e.g. [1, 3, 5] the exchanging of aerodynamic loads still remains unsolved problem. In this work, new fluid to structure scheme is developed and described in details.

An aeroelastic solver has been validate on standard AGARD 445.6 wing [2] and results are presented.

Key words: aeroelasticity, flutter

1. Introduction

Aeroelastics deals with influence of flow on flexible body. Interaction between aerodynamic forces and inertial forces in aeroelastic systems is a well known problem. In such systems aerodynamic forces lead to deformation of the body, and the deformations cause flow conditions and then aerodynamic loads.

Basically, there are two approaches in computational aeroelastics. The first one, joins flow and structural equations into one system of equations. But structural stiffness matrix is approximately one order more stiff than the flow matrix, thus solving global system of these equations is very complicated. Practically, this method is used only for 2D systems.

The second approach in computational aeroelastics, and presented here, is to couple independent programs, treated as modules, into one tool. This strategy involves difficulties with exchanging data between different modules.

Problems occur because aeroelastic model consists of two different models: fluid and structural. Specificity of the issues cause that the meshes are completely different. Although, lots of tools for managing stream of data exist, but exchanging aeroelastic forces between unmatched meshes remains a problem. This paper shows one simple solution of this problem.

2. Aeroelastic Tool

During design process of the aeroelastic tool, several assumptions have been made. Because the tool has a modular structure, no changes in source codes of fluid and structure solvers have been assumed. To simplify the problem, in current approach only small displacements of the structure have been assumed. All modules presented in aeroelastic tool have been designed by authors except a fluid code which can be any arbitrary code. In present work TAU code from DLR has been used as a fluid solver. It is a three dimensional, parallel, hybrid, multigrid code based on finite volume scheme for solving Reynolds-averaged Navier-Stokes equations with wide variety of turbulent models.

As a CSD code, universal solver called MF3 [4] has been used. The solver has a possibility to solve static, dynamic and eigenvalue problems. Its time integration scheme is based on well-known Newmark method and for damping the Rayleigh model is used.

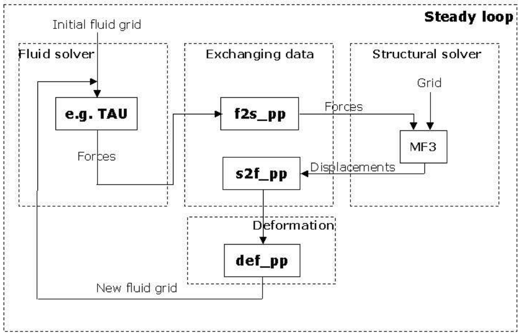


Figure 1. Scheme for steady case calculations.

Generally, there are two kinds of aeroelastic calculations. The first one is a static aeroelasticity where both solvers calculate a steady state solution. The second one is a dynamic aeroelasticity where both solvers calculate time dependent solution. Scheme of the aeroelastic tool for steady state case is presented in Fig. 1. Different modules connected with a special manner can be seen. The modules are the following: fluid (e.g. TAU) and structural solver (MF3), fluid to structure tool (f2_spp) for exchanging aerodynamic forces, structure to fluid tool (s2f_pp) for exchanging structural deflections to fluid grid, and tool for deformation fluid grid (def_pp). First, flow solution have to be calculated for obtaining aerodynamic forces. Next, the forces are transferred from fluid mesh to structural mesh. After calculating deflections of the structure they have to be transferred to deformation tool. Then, computations are restarted with previous solution but on a new grid. In order to speed-up calculations, the coupling process can be done without obtaining converged solution in flow solver in each coupling step. Calculations are stopped when both solvers have converged and fluid mesh does not change.

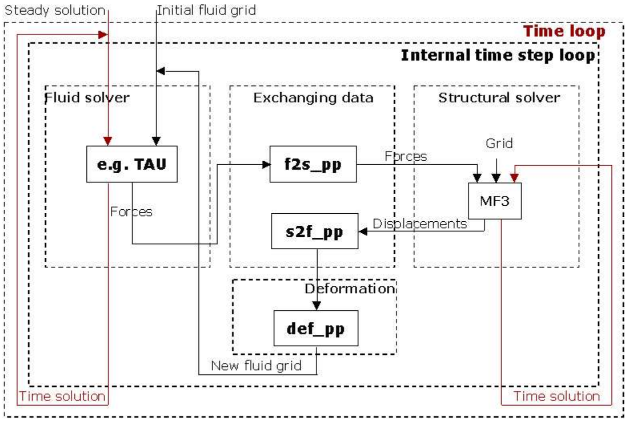


Figure 2. Scheme for unsteady case calculations.

In case of dynamic aeroelastic calculations (Fig. 2), there is an additional time loop that controls stream of data between time steps. Unsteady computations are started from calculated previously steady state solution. Each time step is calculated until both solvers converged to the steady state solution and fluid grid does not change. Several coupling steps have to be done in time step, because only this guarantees that time step really converged. As an initial condition for simple prediction usually the first bending mode is taken. Normally, during unsteady computations, also velocities and accelerations should be transferred. But in the present work, these data are calculated directly from the deflections by flow solver.

As it was mentioned previously, problems occur when aerodynamic forces are transferred between unmatched grids, especially when structural and fluid models are completely different. In that case mesh-based interpolation methods failed to converge because distance between meshes is too big and lots of data are lost. If a zone between meshes is assumed to be a rigid body, then basic mechanical laws can be adopted as a coupling process. Assuming that each aerodynamic force has to be transferred to the close structural element, system of forces on a structural mesh should be equivalent to the system of forces on a fluid mesh.

$$\vec{M} = \vec{r} \times \vec{F}_{fluid}, \tag{2.1}$$

$$\vec{P} = \vec{F}_{fluid}. \tag{2.2}$$

Generally, by moving a force along its direction the most complicated case occurs when common point does not lie on any element. In that case our scheme works as follows. First, aerodynamic force is reduced to the node of the element using equations (2.1) and (2.2) (Fig. 3). The most important thing here is that the edge opposite to the node should puncture plane of work of the force.

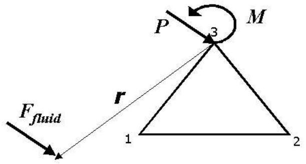


Figure 3. Reduction aerodynamic force to a node

Then, moment $\vec{M}$ is converted into the moment of pair forces $\vec{P}_3$ (see Fig. 4 and (2.3)):

$$\vec{M} = \vec{d} \times \vec{P}_3. \tag{2.3}$$

Vector $\vec{d}$ on Fig. 4 lies on common line of the working plane of the moment $\vec{M}$ and area of the element.

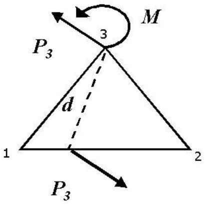


Figure 4. Converting moment $\vec{M}$ into moment of pair forces $\vec{P}_3$.

Because one of the forces $\vec{P}_3$ does not lie on a node it has to be reduced to nodes 1 and 2 (see Fig. 5). Assuming that:

$$\vec{P}_1 + \vec{P}_2 = \vec{P}_3 \tag{2.4}$$

it can be proved:

$$\vec{r1} \times \vec{P}_1 + \vec{r2} \times \vec{P}_2 = \vec{d} \times \vec{P}_3. \tag{2.5}$$

Thus, using equations (2.1) – (2.5) and assuming that nodal forces are perpendicular to the surface of the element nodal forces can be easy calculated.

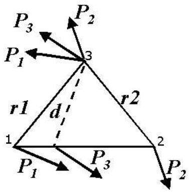


Figure 5. Splitting force $\vec{P}_3$ into nodal forces.

To exchange deflections there have been assumed that each fluid node has the same position in local coordinate system belonging to the most close situated structural element. This method is very quick and sufficiently accurate for small deflections.

For fluid grid deformation, modified spring method has been used. It is so called dual-zone approach, where grid is divided into two zones: elastic and rigid.

3. Test Case: AGARD 445.6 Wing

For validating aeroelastic tools a lot of test cases exist. The most known is an AGARD 445.6 wing [2]. In the current work, the solid model has been used.

As a criterion of compatibility of numerical and reference models, the equality of first four eigenfrequencies and similarity of first four eigenmodes have been taken. Details of computations are presented in Tab. 1.

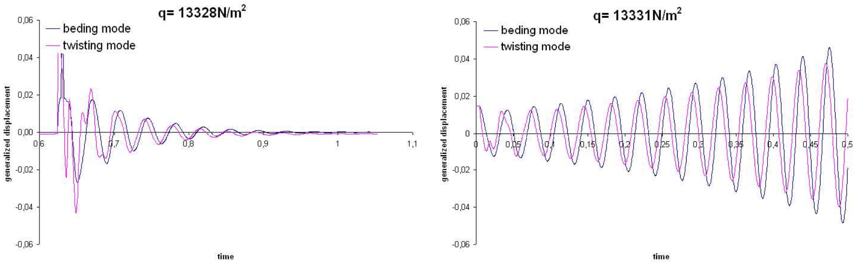
Table 1. Reference and current approach eigenefrequencies of AGARD 445.6 wing

Frequency no.	Reference approach [Hz]	Current approach [Hz]
f1	14.120	14.258
f2	50.913	51.176
f3	68.942	68.607
f4	122.256	121.814

Aeroelastic computations have been carried out with angle of attack $\alpha = 0^o$ at Mach number $Ma = 0.451$ using Euler equations. As a gas, air has been taken. By changing density of the gas, dynamic pressure has been changed and thanks to this flutter point could be determined.

3.1. Results

For monitoring time history of the wing, one node on the tip of the wing has been chosen. On Fig. 6 two cases at different dynamic pressures are presented. The first one presented on Fig. 6a is a case at flutter conditions obtained from experiment. It can be seen that flutter does not occur and system is stable. But if dynamic pressure is slightly increased (see Fig. 6b) the flutter occurs and system became unstable.



a) b)

Figure 6. Time histories at different dynamic pressures.

Using reference [2] a reduced flutter velocity has been computed, it is equal to 0.453 Experimental reduced flutter velocity was 0.488, so the error is about 7.2%. Also an error of exchanging aerodynamic forces has been measured. It is approximately equal to 0.85%.

4. Conclusions

A very simple and quick aeroelastic solver has been developed for flutter simulation. The tool and method presented in this paper are useful and have good accuracy. The flutter prediction has given good comparison with the experimental data. In the future, aeroelastic tool will be enhanced for big displacements cases and nonlinear materials.

References

[1] R.L. Harder and R.N. Desmaris. Interpolation using surface splines. *Journal of Aircraft*, **9**(2), 189 – 191, 1972.

[2] E.C. Yates Jr. *AGARD standard aeroelastic configurations for dynamic response I.-Wing 445.6.* AGARD Report No. 765, 1988.

[3] F. Liu, J. Cai, Y. Zhu, H.M. Tsai and A.S.F. Wong. Calculation of wing flutter by coupled fluid-structure method. *Journal of Aircraft*, **38**(2), 334 – 342, 2001.

[4] M. Morzynski and M. Nowak. Generic structural tool for aeroelastic analysis. In: *Conference Proceedings, September 7-9, 2004*, 45-th International Conference of Machine Design Departments, Blansko, Czech Republic, 72 – 77, 2004.

[5] M.J. Smith, D.H. Hodges and C.E.S. Cesnik. Evaluation of computational algorithms suitable for fluid-structure interactions. *Journal of Aircraft*, **37**(2), 282 – 294, 2000.