Mathematical Modelling and Analysis 2005. Pages 97–102
Proceedings of the 10th International Conference MMA2005&CMAM2, Trakai
 ISBN 9986-05-924-0

FLUID-STRUCTURE INTERACTION IN SIMULATION OF TRANSONIC FLOWS

Ł. JEZIOREK and J. ROKICKI

Institute of Aeronautics and Applied Mechanics

Nowowiejska 24, 00-660, Warsaw, Poland

E-mail: [jeziorek,jack]@meil.pw.edu.pl

Abstract. The elasticity of wing has a large influence on the flight performance of an aircraft. This paper presents fully coupled aerodynamic and elastic simulation for the case of transonic flow around large transport aircraft. The flow solver is based on the compressible viscous-inviscid interaction method. The structure elastic displacement is obtained from MSC NASTRAN^{TM}code.

Key words: aeroelasticity, transonic flows

1. Introduction

In practical numerical simulations of transonic flows elastic deformation of structure may influence the flow in strongly nonlinear manner. In particular this is the case for the swept wing for which the lift force bends the wing the same time twisting it along the span. The bending itself is a second-order effect, the twist however modifies the local angle of attack, strongly influencing the flow (and especially the pattern of shock waves). This effect can be observed on the aircraft in free flight, as well as in the wind tunnel model [1, 2, 3, 4]. In the latter case the structure is quite rigid (the model is made out of solid steel), yet the aerodynamic pressure might be very high in order to artificially increase the Reynolds number. As a result the additional wing twist angle may reach locally near the wing tip 1-2 degrees. Such a change of geometry is sufficient to modify significantly the outer flow and the aerodynamic forces themselves.

The paper presents the computational method implementing strong coupling between three elements:

- the solver of the Euler equation simulating inviscid compressible flow (finite volume discretization on external unstructured tetrahedral mesh),
- the solver of the 3D boundary layer equations (finite-difference discretization) describing viscous effects,

- the structural analysis tool predicting the elastic deformation (finite element method on internal hexahedral mesh).

2. Finite Element Model of the Wing

In the wing-body geometry the Finite Element (FE) model represents the wing only. All displacements on the wing root are assumed zero. Loads are prescribed as piecewise constant pressure on each FE surface. The wing is assumed to be made of a solid steel (which is a good approximation of the wind-tunnel model used in experiments). The deformation is predicted using MSC NASTRAN^{TM}simulation tool.

3. Finite Volume Flow Solver

3.1. Flow solver

The in-house HADRON code implements a viscous-coupled method consisting of the following elements:

- Inviscid Euler solver.
- Boundary Layer solver.
- Coupling method.
- Inter-grid interpolation.

3.2. Inviscid Euler solver

The Euler solver bases on the conservative, finite volume, cell-centered approach. It accepts tetrahedral unstructured meshes in the defined apriori format. The pseudo-time marching method is used together with the implicit time-discretization. The resulting linear system is solved using either block Gauss-Seidel or Bi-CGSTAB methods. The inviscid fluxes are calculated using Warming-Steger/Van-Leer flux vector splitting methods. On the cell walls the conservative variables are calculated with enhanced accuracy by the second-order WENO (Weighted Essentially Non-Oscillatory) scheme.

3.3. Boundary Layer solver

The flow inside the boundary layer is considered to be fully three dimensional, compressible, laminar or turbulent, attached or separated. The method uses a surface-oriented non-orthogonal coordinate system for the boundary layer equations, i.e. continuity and momentum equations. These equations are solved in transformed, dimensionless form using finite differences method basing on predictor-corrector scheme. Instead of energy equation, the Crocco integral is used. The density distribution is calculated from the equation of state.

To close the system of boundary layer equations, the eddy-viscosity concept is used which relates the Reynolds shear stresses to mean velocity profiles. The eddy-viscosity coefficient is calculated using different formulas in the inner and outer regions of the boundary layer. The boundary layer development in the wake is calculated by the 2D Green' s integral method.

After solving the boundary layer equations separately for upper and lower surfaces, the transpiration velocity related to the displacement thickness of boundary layer is determined. Finally, calculations of the wake shear layer using two-dimensional integral Green's method in chordwise direction are performed.

3.4. Flow coupling method

In the viscous/inviscid interaction technique, inviscid solver is coupled with a boundary layer code using Le Balleur' s method. This method distinguishes between direct and inverse boundary layer computational modes which are used for attached/separated flow regimes, respectively. Grids used by the Euler and Boundary Layer solvers are different. The former is tetrahedral unstructured and the latter is structured and hexahedral. These grids coincide approximately on the wing and wake surfaces, but even there the grid nodes have different location. In order to implement the coupling procedure the following steps were implemented:

- The unstructured surface mesh covering the wing and the wake mesh are extracted
- The interpolation procedure was established to transfer flow data from the cell centers to the surface grid nodes (of the unstructured grid).
- Another interpolation procedure was established to redistribute the transpiration velocity (obtained in the reference frame of the structured boundary-layer grid) to the centers of the surface cells of the unstructured Euler grids (on the wing and in the wake).

4. Coupling of the CFD and CSM Solvers

The aeroelastic coupling between CFD and CSM solvers is obtained by the iterative procedure consisting of the following steps (see also Fig. 1):

1. The undeformed CFD grid is taken for the first flow simulation.
2. The flow solver calculates pressure field on the wing using most recent version of the CFD grid.
3. The obtained pressure field is reinterpolated on the CSM grid.
4. The CSM solver (MSC NASTRAN^{TM}) uses the pressure loads to calculate the displacement of the wing surface.
5. The obtained displacement field is reduced to the pure shift and twist of the planar wing cross section.
6. The original CFD grid is distorted to accommodate the estimated wing displacement.

7. If the calculated displacement is sufficiently small, the loop is terminated, otherwise the control returns to step 2.

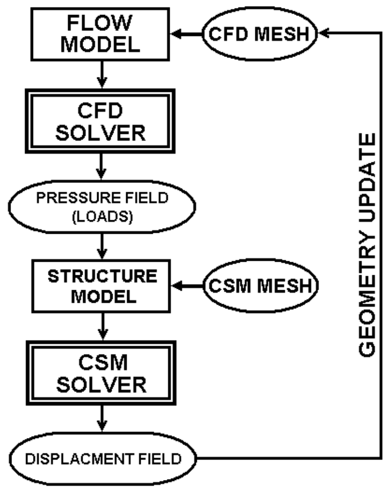

Figure 1. Scheme of computational coupling.

4.1. Wing deformation model

The surface displacement field obtained from CSM analysis is too detailed to be useful in CFD flow simulations. Additionally it would be quite difficult to propagate such a general surface displacement field onto the spatial CFD mesh. The following assumptions are made:

- All wing profiles (y=const cross section) are planar and rigid.
- Displacement is reduced to:
  - horizontal translation ΔX,
  - vertical translation ΔZ,
  - twist angle $\Delta\phi$ of the profile.
- Both translations and local twist angle are functions of the horizontal coordinate only ($\Delta X(y)$, $\Delta Z(y)$, $\Delta\phi(y)$).

It must be noted that in present applications $\Delta X(y)$ is negligible while $\Delta\phi(y)$ reaches at most few degrees. The vertical translation may be significant, yet its influence on the flow is expected to be small. In practice the functions ΔX, ΔZ, $\Delta\phi$ are calculated with a very good result from the surface displacement field using the least square approximation.

4.2. Deformation of the CFD mesh

Many possibilities exist to propagate local bending and twist to the whole spatial CFD mesh. Here, the very simple method is presented, which bases on

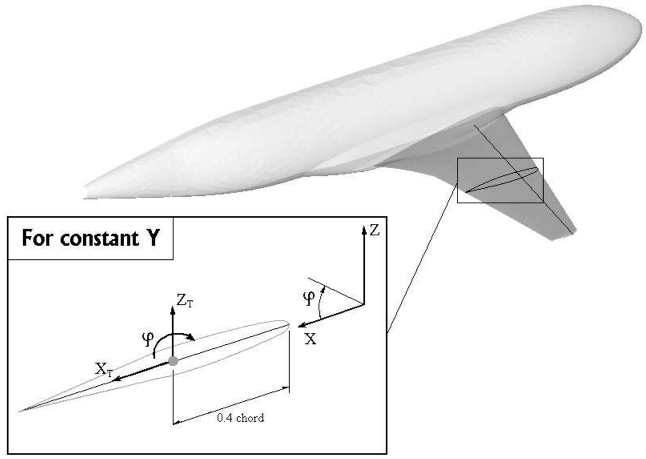

Figure 2. Description of deformation.

the algebraic transformation. At first we shall observe that both $\Delta\boldsymbol{r}(y) = \Delta X(y) + i\Delta Z(y)$ and $\Delta\phi(y)$ are prescribed on the whole wingspan for $y_{WR} < y < y_{WT}$, where y_{WR} stands for the wing-root position (fuselage thickness) while y_{WT} is the y-coordinate of the wing tip. In agreement with the deformation model mentioned in Section 2:

$$\Delta\boldsymbol{r}(y_{WR}) = 0, \quad \Delta\phi(y_{WR}) \equiv 0.$$

All three functions can be continuously extended for $y > 0$ in the following manner:

$$\widetilde{\Delta}\phi(y) = \begin{cases} 0, & 0 < y < y_{WR}, \\ \Delta\phi(y), & y_{WR} < y < y_{WT}, \\ \Delta\phi(y_{WT}), & y > y_{WT}, \end{cases}$$

the formula for $\widetilde{\Delta}\boldsymbol{r}$ is analogous. The global transformation has the following form:

$$\boldsymbol{r}_{\mathrm{NEW}}(y) = \left(\boldsymbol{r}_{\mathrm{OLD}}(y) - \boldsymbol{r}_{\mathrm{A}}(y)\right) e^{i\widetilde{\Delta}\phi(y)} + \boldsymbol{r}_{\mathrm{A}}(y) + \widetilde{\Delta}\boldsymbol{r}(y),$$

where $\boldsymbol{r}_A(y) = x_A(y) + i\, z_A(y)$ stands for the coordinate of the rotation axis.

5. Numerical Results

The algorithm described in previous Sections was applied in the case of transonic flow (Ma=0.85, $\alpha = 1.0°$) around the wing-body geometry presented in Fig. 2. Figure 3 presents the convergence of the iterative procedure for the span-wise distribution of vertical translation and twist angle.

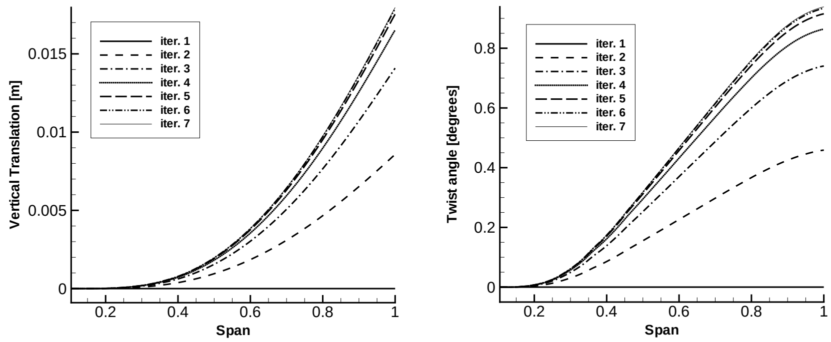

Figure 3. Vertical translation and twist angle along span.

6. Conclusions

The coupling algorithm presented in the paper has proved to be convergent for transonic flow where interaction is highly nonlinear. The results show large wing deformations (twist and vertical deflection), despite the fact that the structure was made of solid steel. In Fig.3 the maximum value of twist angle is about 1 degree. Such large geometry alteration causes extensive modification of the shock placement, especially for the outer part of the wing. This may change both drag and lift and affect performance of the wing.

References

[1] Armin Beckert. Coupling fluid (CFD) and structural (FE) models using finite interpolation elements. *Aero. Sci. Technol.*, **4**, 13 – 22, 2000.

[2] Manoj K. Bhardwaj. *A CFD/SCD Interactions Methodology for Aircraft Wings.* Virginia Polytechnic Institute and State University, PhD dissertation, 1997.

[3] Guru P. Guruswamy. A review of numerical fluids/structures interface methods for computations using high-fidelity equations. *Computers and Structures*, **80**, 31 – 41, 2002.

[4] Jamshid A. Samareh. Multidisciplinar aerodynamic-structural shape optimization using deformation (massoud). *AIAA Paper 2000-4911*, 1, 1 – 11, 2000.