Aeroelastic Simulation for Symmetric Manoeuvre of I23 Manager Light Plane

Robert Roszak\textsuperscript{1} Witold Stankiewicz\textsuperscript{1}, Marek Morzyński\textsuperscript{1}, Wojciech Chajec\textsuperscript{2}

\textsuperscript{1}Institute of Combustion Engines and Transportation, Poznan University of Technology  
Robert.Roszak@put.poznan.pl,  
\textsuperscript{2}Institute of Aviation, Warsaw, chajec@rzeszow.home.pl

Abstract

Expansion of computer technologies allow using numerical simulation in the early stages of aircraft design more and more often. The role of both wind tunnels and initial test flights used to verify the validity of solutions seems to be diminishing. Big systems for three-dimensional simulations of Fluid-Structure Interactions (FSI) constitute highly specialized and costly software. Most of the codes are based on many simplifications. In this paper fluid-structure interaction, taking into account the symetric manoeuvre of ultra light plane (fig 1.), is concerned. This phenomenon has important influence in many aeronautical applications. The method and developed system is demonstrated on ultra light I23 plane. For the first flow the comparison with experiment made in Institute of Aviation Warsaw is presented. Finally, aeroelastic simulation of full I23 aircraft configuration presents the capability of used numerical codes to analyze largescale complex geometries for manoeuvre. All computations were carried out in parallel environment for CFD mesh of order of millions tetrahedral elements.

1. Introduction

Expansion of computer technologies allow using numerical simulation in the early stages of aircraft design more and more often. The role of both wind tunnels and initial test flights used to verify the validity of solutions seems to be diminishing. Big systems for three-dimensional simulations of Fluid-Structure Interactions (FSI) constitute highly specialized and costly software. Most of the codes are based on many simplifications. One of them is the assumption of linearity of the structural model being in contradiction with real-life situations. The paper presents the results of simulations for complex, multi-scale object models – I-23 Manager. What is crucial for carrying out the assumed analyses is to extend a numerical tool [1] comprising a flow and a structural program and a space grid deformation model for a system. The scope of our work has included:

- Joining independent programs: flow, structural, interpolation and three-dimensional CFD grid deformation tools into one integrated system,
- Adapting a structural code to a non-linear analysis and modifying FSI control script,
- Carrying out tests,
- Analyzing FSI on certain examples,
- Visualizing the results.

The point of reference for testing the suggested approaches is the existing solutions of the aeroelastic linear problems. The paper is organized as follows. In section 2 the brief description of Computational Aeroelasticity problems are presented. The methodology of Fluid-Structure Interaction is given in section 3. Finally the developed and validated algorithm is demonstrated on full I-23 aircraft configuration.

2. Computational Aeroelasticity

Computational Aeroelasticity [2] is a branch of mechanics, which examines the way a stream of fluid affects a deformable body that it flows around. The term combines methods used in Computational Fluid Dynamics (CFD) and Computational Structural Mechanics (CSM) [3]. The non-linearity found in structural models is usually not taken into consideration in numerical analyses of interacting fluid and structures found in the literature [2, 3, 4, 5]. This limits the possibility of simulating such cases as manoeuvres of planes, in which considerable deformation of structure occurs, and cases with the non-linearity of constitutive equation.

The non-linearity is also of particular importance for biological flows as for example blood flow in the blood vessels [5]. Because of the wide scope of FSI, the paper is focused on fluid and structure interacting in external flows. Computational methods for different aspects of aeroelastic responses are still the subject of scientific examination. To illustrate the point, many aspects [2] in are FSI related to searching interdependence between aerodynamics in a flow and the Dynamics of structure. This approach is connected with many complications stemming from two independent numerical codes interacting with each other.

Another problem to be solved is the way of exchanging information between programs for fluids and structure (figure 2.1), with a different level of the discretization and a use of different methods (finite elements, finite volumes). One significant problem in analyses of FSI is a description of particular states in relation to two different coordinate systems used. What is used quite often is [2] use of Euler’s description - a stationary coordinate system (CSM) or Lagrange’s description - a movable coordinate system (CFD). Thus it is necessary to devise appropriate techniques of data exchange between these two systems. Figure 2-1 is an example of differences in
discretization for the NACA 0012 airfoil: a) a CFD grid - tetrahedral elements, b) a structural grid - beam and surface elements. What also poses a problem is adopting a time step in calculations which can be different for a flow code and for a structural code. It is particularly important when conducting a dynamic analysis of a FSI.

What also poses a problem is adopting a time step in calculations, which can be different for a flow code and for a structural code. It is particularly important when conducting a dynamic analysis of a FSI.

3. The Fluid Structure Interaction Algorithm

There are many methods to perform fluid-structure interaction computations. In the first approach, equations describing all the coupled physical phenomena are coupled in one code. As each component of the coupled problem has different mathematical and numerical properties (linear/non-linear equations, symmetric/unsymmetric matrices, etc.) [7], this approach is computationally challenging.

The alternative approach, used in this work, is based on the code coupling, where fluid flow computations and structural analyses are performed by separate packages. Depending on the particular problem, different numerical strategies might be employed. The overview of them might be found in pioneer studies by Farhat, Piperno and others [8, 7, 9]. To carry out aeroelastic calculations a control system has been created which has interrelated particular numerical tools into one integrated system. The designed control scripts allow efficient calculations. The program starts with static analysis, then the perturbation of the solution is introduced and dynamic analysis continues. The following tools described in the subparagraphs 3.1-3.4 have been used in the presented system.

3.1 TAU-Code.

For CFD computations, a parallel and efficient RANS flow code (Reynolds Averaged Navier–Stokes) by Deutsches Zentrum fuer Luft- und Raumfahrt (DLR) [10], has been used. The system consists of two modules: the former is used to prepare a task for calculations (pre-processing), the checking of the grid, the division into subdomains (in the case of parallel calculations), the introduction of boundary conditions, etc.; the latter solves a system of equations (Euler’s or RANS if turbulence and viscosity are taken into account).

3.2 The deformation module.

It is a module for modifying a CFD grid on the basis of the deformation of the structure; the aim of this modification is to change the CFD grid in such a way that the points (nodes) of the coupling surface lie on the surface of the object being flown round, while preserving the quality of the mesh. Multiple methods used for mesh deformation incorporate the spring analogy concept. All of them are based on the assumption, that the tetrahedral (cubic, etc.) elements of the CFD mesh are replaced with spring elements. The difference between them is the number of interpolation triangles that are activated in each case. Among these methods, torsional spring analogy [11, 12], semi-torsional spring analogy [13, 14], ortho-semi-torsional (OST) spring analogy [15] and ball-vertex spring analogy [16] can be mentioned. For the purpose of aeroelastic computations the efficient mesh deformation system has been developed. It is based on in-house MF3 structural code and beam elements. It compromises benefits of spring and torsional orthotropic spring analogy.

3.3 The MF3 structural code.

For elastic calculations MF3 [17] - the in-house finite element code has been used. It allows static and dynamic calculations and a modal analysis using 1-dimensional (beams, rods), 2-dimensional (shells, membranes, etc.) and 3-dimensional (tetrahedral, hexahedral) elements.

3.4 The F2S and S2F modules

The fluid-structure interaction algorithm, used in aeroelasticity analyses, requires the transfer of some quantities between the used codes. In the interaction mentioned before, pressures computed in CFD software act as the loads in structural code. The deformations of structural mesh, resulting from CSM analyses, influence the computational domain and the boundary conditions in the CFD part of the coupled system. The interpolation algorithms are divided to the ones based on the geometry and the ones based on the finite element mesh. The example of the first group is spline interpolation [18, 19]. The codes based on the definitions of mesh points [20, 21] usually define the surfaces, where the interaction between coupled codes occurs. Due to the fact that coupled software is multidisciplinary, and the domain used in structural analysis usually is not rigid, the interpolation tools have to determine how the coupling surfaces of both Romain fits together [21].

There are several different search algorithms finding the adequate pairs of points and elements on the coupling surfaces. One of the most straightforward of them is linear search, shown in [22]. As has been described in [21], the computational cost of the comparison of all pairs of points rapidly increases with the growth of mesh size. Another methods, e.g. oct-tree [23, 24] and bucket [25, 26, 27] search algorithms, are based on restricting the search region and are much more efficient.
The detection strategies differ for matching and non-matching meshes. In the first case, the structure of the both contact surfaces is the same, and only the pairs of points have to be found. For different levels of discretisation or in case when the geometry in one of the coupled tools is simplified, the algorithm for non-matching grids has to be used. In this case, the pairs of points and elements have to be found. Again, bucket and oct-tree algorithms are more efficient than linear search.

When the neighbourhood of the points and elements is computed, the interpolation can be performed. Depending on the type of the quantity to be interpolated, different methods can be mentioned:

- **Non-conservative interpolation** might be used for the functions of spatial coordinates in time, like the pressures, velocities or the mesh points coordinates.

![Non-conservative interpolation](image)

Figure 3.1. Non-conservative interpolation. Data is interpolated from source points \( Q_i \) to a target point \( Q_t \)

In the case of non-conservative interpolation (figure 3.1), for each point of target mesh the corresponding element on the source mesh is found. Next, the values from source points are interpolated to the target point (lying inside the Skurce element).

- **Conservative interpolation** is adequate in the cases, where some additional laws must be preserved during interpolation. The example of such situation is the interpolation of the forces, where their sum has to be preserved. In that case (figure 3.2) the source value is located in single point and transfer red to the points of element lying on target coupling surface, using weights \( w_i \) that satisfy \( \sum w_i = 1 \). Each node on target element gets a portion \( w_i \) of given coupling quantity.

![Conservative interpolation](image)

Figure 3.2. Conservative interpolation. Data is interpolated from a single source point \( Q_t \) to the target points \( Q_i \) at the corners

For interpolation in the system presented here three sets of modules were tested. The first set is the EADS modules developed in frame of TAURUS project [28]. They are based on conservative solutions of finite elements, used to distribute the pressures from the CFD grid to the structural one (forces), and the displacements from the structural grid to the CFD grid. The same goal could be obtained by the second set of modules employing the MpCCI [21] tools. MpCCI-based interpolation was successfully tested in Poznan University of Technology in frame of TAURUS project [28].

The third set of modules is the developed in-house [29]. In the tools used by authors, bucket search algorithm has been used. The comparison of all three sets of interpolating modules show no significant differences in performance and accuracy. They all allow interpolation between non-matching grids. EADS and in-house modules performed better in the cases, when only torsion box of the wing was modelled on the structural side. For further examples shown the EADS modules were used.

### 3.5. The description of the algorithm

Below the simulation algorithm for static FSI calculations is presented (figure 3.3). The static computations are performed to specify initial conditions for a dynamic FSI analysis. In aeroelastic computations the structural model is subjected to the forces determined on the basis of pressure distribution computed with the CFD code. The analysis is started with CFD computation. The process consists of three stages. The first stage is about generating the consistent CFD grid (figure 3.4) and checking it for negative volumes and other grid pathologies. Then the grid is partitioned. The division is dependent on the number of used processors, and in turn this number is dependent on the size of the task and the amount of time for the MPI packet to pass on messages.

![Algorithm](image)

Figure 3.3. The static FSI analysis, left - the starting point for adaptation a CFD grid

The next stage constitutes a significant part of the flow analysis. The boundary conditions having been specified and the environment having been prepared, computations are performed. After the flow analysis, parameters for information exchange are specified (units, the coupling surface, object files). Then the CFD data, in the form of the pressure distribution on the coupling surfaces, are translated into forces, and interpolated from the CFD grid to the structural grid. The exchange occurs through the coupling
surface, specified on the basis of the structural model and flow model. The next step consists of structural calculations by means of structural code. Depending on settings, it is possible to perform a traditional linear analysis or to take into account the constitutive and geometrical nonlinearities. When the structural calculations have been completed, the information in the form of the displacement of given nodes of the structural model is passed on to the deformation module.

On the basis of the structure’s displacement, the CFD grid is modified (figure 3-4), thus creating a new grid for the next step in the calculations. The process of deformation being realized is based on the spring analogy.

The designed tool also enables performing dynamic analyses, which is necessary in analysing dynamic response of the aircraft depending on types of the input function. In the case of dynamic simulations, the algorithm has been extended with an additional iterative loop (figure 3-5).

In this case, after introducing the initial condition (the input function) the dynamic response of the system to the initial perturbation in a given period of time is investigated. Dynamic aeroelastics also takes into account the acceleration of the structural grid (the grid’s dynamics) and its impact on the flow. Information on the dynamics of the grid is obtained on the basis of current displacements of the grid in reference to this obtained in the previous time step as well as on basis of the previous flow solution. Typical time coupling procedures used for aeroelastic simulations are described in [30, 31]. Here the partitioned time stepping is used. The time coupling is solved explicitly as boundary conditions imposed by one system onto the other. To prevent numerical instability the time step is decreased and subiterations are performed. By subiterating the fully coupled solution is obtained. The size of time step for all the used modules is managed by CFD solver TAU-code, interfaced for this kind of aeroelastic computations in TAURUS project [28]. The description of similar management of aeroelastic time coupling can be found in [32].


The structural model of airplane has been obtained by (in frame of TAURUS project and later) Institute of Aviation Warsaw (figure 4-1, left). The model consists of beam and mass elements. The boundary conditions have been specified in line with the data from the Institute of Aviation. To perform the computations the three-dimensional non-structural CFD grid has been generated (figure 4-1, right). The grid consists of about 15 millions of tetrahedral elements.

The aeroelastic system is based on measured eigenmodes adopted from IoA. Examples of these modes are shown on figure 4-2.

Below one can see the results of the calculations of displacement for the control nodes W1 and W2 of the flow model (figure 4-3) in the function of time.


The structural model of airplane has been obtained by (in frame of TAURUS project and later) Institute of Aviation Warsaw (figure 4-1, left). The model consists of beam and mass elements. The boundary conditions have been specified in line with the data from the Institute of Aviation. To perform the computations the three-dimensional non-structural CFD grid has been generated (figure 4-1, right). The grid consists of about 15 millions of tetrahedral elements.

The aeroelastic system is based on measured eigenmodes adopted from IoA. Examples of these modes are shown on figure 4-2.

Figure 4-2. Examples of structural modes for I23 Plane, a) mode 1, b) mode 2 and c) mode 3

Below one can see the results of the calculations of displacement for the control nodes W1 and W2 of the flow model (figure 4-3) in the function of time.


The structural model of airplane has been obtained by (in frame of TAURUS project and later) Institute of Aviation Warsaw (figure 4-1, left). The model consists of beam and mass elements. The boundary conditions have been specified in line with the data from the Institute of Aviation. To perform the computations the three-dimensional non-structural CFD grid has been generated (figure 4-1, right). The grid consists of about 15 millions of tetrahedral elements.

The aeroelastic system is based on measured eigenmodes adopted from IoA. Examples of these modes are shown on figure 4-2.

Below one can see the results of the calculations of displacement for the control nodes W1 and W2 of the flow model (figure 4-3) in the function of time.

Figure 4-3. A Euler flow grid with control nodes for displacement and rotations analysis.
4.1. Manoeuvre analysis

The normal modes of structural model has been obtained too in cooperation (in frame of TAURUS project and later) with the Institute of Aviation Warsaw (IoA) (figure 4.1, left). The model consists too of mass elements and dummy beams. The mode shapes were obtained based on the ground vibration tests results. The boundary conditions and parameters for manoeuvre analysis have been specified in line with the data from Institute of Aviation. The grid generated in this way has been subjected to quality tests. The FSI has been carried out for the following boundary conditions and parameters:

Velocity $V_x = 69.44 \ [\text{m/s}]$
Atmospheric pressure $P = 0.1 \ [\text{MPa}]$
Reynolds number $Re = 2 \cdot 10^6$
Angle attack = 1.76 (start parameters)
Time step $t = 0.01 \ [\text{s}]$
Full input parameters shown in work from IoA.

The FSI analyses of the I-23 has been carried out for both, static and dynamic conditions. The result of the static FSI analysis was input into the dynamic one. The perturbation is introduced as the impulse force on the wing tip. The results of the analysis are shown in the following figures.

The aeroelastic system base on vibrations mode from modal analysis of structural model. For manoeuvre calculation adding two rigid body modes (Fig. 4-4).

Below one can see the displacement „Z-axis” and rotation „Y-axis” for structural (figure 4.5 left) and flow model of the I-23-Manager plane.

![Figure 4-4. Rigid body mode shapes for maneuver analysis.](image)

![Figure 4.5. Displacement and rotation of model I-23 plane: CSM model (left) and CFD model (right).](image)

On figure 4-7 shown the position for model I-23 Plane on time steps.

![Figure 4.6. Rotation „Y-axis” of I-23 plane.](image)

![Figure 4.7. Steps of the position I-23 plane on time steps.](image)

4.2. Results of calculations

Below one can see the results of the calculations of displacement „Z-axis” and rotation „Y-axis” for manoeuvre analysis (figure 4-5) in the function of time. The displacement „Z-axis” is defined like trajectory of the I-23 – manager. On left side shown the trajectory who was theorectical calculated by W. Urbaniak from IoA.

![Figure 4.4. Trajectories of the I-23 maneuver: theoretical calculations (left) and Fluid Structure Interactions analysis (right).](image)

5. Conclusions

The I-23 plane’s analysis shows that the presented here numerical tool can be used in design process in the aviation industry as well as in examining dangerous phenomena, such as the flutter limit cycle oscillation (LCO). In case manoeuvre analysis the
results of numerical calculations are very similar like provided by Mr. Urbaniak from IoA (Work In Cesar). This code can be used to calculations of real objects with complex geometry – like I-23 – Manager. This code is ready for calculations of the manoeuvre.

6. References


