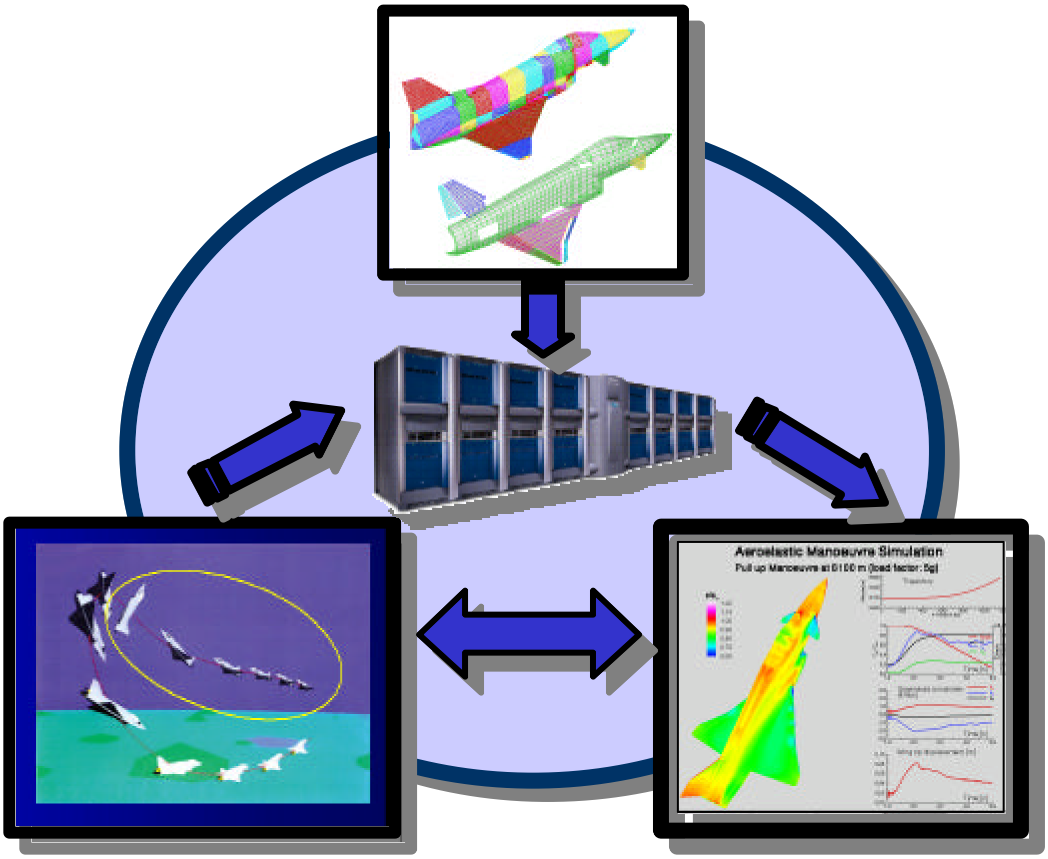
AIAA 2002-0949

Time-Dependent Aeroelastic Simulation of Rapid Manoeuvring Aircraft

L. Fornasier, H. Rieger, U. Tremel and E. van der Weide

EADS-M

Ottobrunn, Germany



40th AIAA Aerospace Sciences Meeting & Exhibit
14-17 January 2002
Reno, Nevada

TIME-DEPENDENT AEROELASTIC SIMULATION OF RAPID MANOEUVRING AIRCRAFT

L. Fornasier*, H. Rieger†, U. Tremel‡, E. van der Weide‡

EADS –M , Flight Physics,
Ottobrunn, Germany

Abstract.

This paper presents the aeroelastic-related computational activity performed by the EADS-M team in the ESPRIT Project EP25050 JULIUS. The main objective of the JULIUS project was the development of 6S, a Problem Solving Environment, designed to provide an integration platform for software modules required to simulate complex, large scale, industrial problems. As part of the industrial demonstrator suite defined to assess 6S capabilities when performing large scale, multi-physics computational simulation, EADS-M has simulated the fluid-structure interaction during rapid aircraft flight manoeuvres at transonic conditions by use of an advanced, time-accurate CFD-CSM coupling methodology. The aerodynamic modelling, elastomechanical modelling and the temporal and spatial coupling algorithms implemented in this approach are presented. The solution of the unsteady Euler equations for transient flow involving moving boundaries for realistic 3-D configurations is computationally expensive. Use of highly parallel computers is one of the ways for reducing the run times. The design of the coupling procedure and its components has been driven by computational efficiency requirements on parallel, distributed memory architecture machines, resulting in a highly efficient implementation which can utilise today's high performance computing resources. The paper addresses the computational issues. Finally, the results obtained from the simulation of a 5-g pull-up manoeuvre for a generic X-31-Experimental Aircraft Configuration are presented.

Nomenclature

C_D	aerodynamic drag coefficient
C_L	aerodynamic lift coefficient
p/p_∞	dimensionless pressure coefficient
q	generalised co-ordinate

Introduction

Traditionally, computational aeroelasticity for prediction of loads and flutter combines a linear finite element formulation for the structure with linear aerodynamic methods as the doublet-lattice method or panel method on a simplified representation of the aircraft configuration. At the same time, prediction of 'flexible' aerodynamic performance and control surface effectiveness is done by taking into account the effects due to the structural elastic deformations on the external aerodynamics by means of 'correction' factors applied to the 'rigid' data. Both practices are well-established in aircraft design and – when properly applied - give accurate, reliable and rather inexpensive predictions for static and dynamic effects at subsonic and supersonic speeds. In the transonic flight regimes and for rapid manoeuvring conditions - where non-linear aerodynamic effects may become not negligible – the aircraft design, development and certification processes have to rely on expensive flutter and aerodynamic experimental models and on extensive flight testing. Adoption of innovative, unconventional designs, aerodynamically unstable configurations and automatic Flight Control Systems (FCS) in modern aircraft exacerbate further the presence and the impact of aerodynamic non-linearities. As reported in [1], most of the recent U.S. aircraft programmes have experienced problems due to inaccurate prediction of aerodynamic loads that involve non-linear phenomena, such as shocks, vortices and separated flows. Examples includes adverse aeroelastic effects as increased trim drag and loss of control effectiveness, transonic flutter, Limit Cycle Oscillations (LCO) , control surface buzz and fin buffet. Access to the recent advances in non-linear Computational Fluid Dynamics (CFD) is felt as a high potential for increasing the accuracy of the prediction methodology for structurally flexible aerostructures in flight regimes involving non-linear aerodynamic phenomena. A pilot implementation of the modelling of a time-accurate, non-

* Project Manager Flight Physics, AIAA Member
† Manager Numerical Simulation
‡ R&D Engineer Numerical Simulation

linear fluid-structure interaction in the time-domain has been carried out within the ESPRIT Project EP 25050 JULIUS. A short description of the main features of the simulation system and an overview of the results from its application to the aeroelastic simulation of a manoeuvring aircraft are presented in the following.

Background

Computation of fluid-structure interaction using non-linear aerodynamics has been done in more or less sophisticated ways since the '80. In the earlier implementations, a modal representation of the elastic motions has been coupled with flow solvers solving the transonic small perturbation equations (TSP) or the full potential equations (FPE) by finite difference schemes (FDE) for isolated wing configurations and strongly simplified full aircraft geometry. Structural deformations were mainly treated by equivalent transpiration boundary conditions. Today's improved capabilities in disciplinary modelling and computational resources make possible to directly couple in the time-domain viscous, time-dependent transient Navier-Stokes flow solvers with structural analysis methods for realistically complex, full aircraft configurations.

However, coupling of a time-dependent flow solver solving the transient, compressible equations for inviscid (EULER) or viscous (Navier-Stokes) flows about a complex aircraft geometry to a structural analysis method is not a trivial task and requires efficient solution of several physical and computational issues not present in the 'classical' linear aeroelastic technology.

Modern flow solvers rely on a Finite Volume (FV) formulation which requires the regions around the aircraft to be sub-divided into a mesh, which consists of a large number (millions) of elementary volumes in which simplified forms of the governing equations can be solved. The generation of such meshes around a realistically complex aircraft geometry is – per se – a challenging task. During flight manoeuvre simulations, the aircraft surfaces – i.e. a subset of the mesh boundaries – move as result of rigid body translations and rotations, relative local motion – e.g. flap deflection – and structural elastic deformations. Due to the presence of the moving boundaries the shape of the computational domain changes continuously and a deformation algorithm is required to adapt the internal elements. The actual shape of the configuration is determined by the interaction between aerodynamic, elastic, gravity and inertial forces. The simultaneous determination of all forces in one model is restricted to analytical treatment of simple problems.

For complex problems, computer-based solution methods are required which make the use of a loose-type coupling more attractive. In a loose-type coupling the different forces are computed in a 'concurrent' fashion in the different physical domains, e.g. the aerodynamic forces are computed by the CFD code independently from the elastic forces computed by the structural analysis code and vice versa. The equilibrium of all forces is achieved by exchanging information between the different disciplines. As the aerodynamic and the structural code operates on each own computational grid – in space and time – the exchange of information – both in space and time – has to be consistent. This duty is taken over by an aeroelastic steering module, where two coupling operators are defined: a spatial operator, which transfers the loads from the aerodynamic simulation to the structural model and map the structural deformations onto the CFD mesh boundaries, and a temporal operator that yields a time-consistent equilibrium between all the forces acting on the system. It is noteworthy to remark that within this logical scheme the static aeroelastic case can be considered as the final equilibrium-state with zero-inertial forces case of the more generic transient formulation.

As it is well known, the accurate resolution of transient, non-linear aerodynamics about complex geometries is computationally expensive. Massively parallel computers have successfully been used to reduce the run times of such simulations. The information exchange in the fluid-structure coupling, the treatment of the moving boundaries, and the presence of the structural model largely increase the complexity of the computational model. Special attention must be used to preserve computational efficiency on highly parallel computers, which requires parallelisation of all the time-consuming parts of the simulation sequence and efficient transfer of data between the different discipline across the processors.

Clearly, the whole sequence of operations needed to set up and steer the coupled simulation can be configured by the user via shell scripts. However, in our opinion, the inherent complexity of such a system is much more efficiently resolved by an object-oriented (OO) approach. Through the OO concepts of polymorphism, inheritance and encapsulation ([2], [3]) the building of large and complex software systems is well supported. Compared to the procedural programming used with Fortran77 or C adoption of an OO approach results in better readable, maintainable and extensible code, through the powerful features supported in OO languages like strongly typed interfaces, templates, design patterns. ([4], [5]).

The SimServer Environment

Basic Concepts

The simulation of the aircraft manoeuvre described in the next section has been performed using the SimServer-system developed by the third author within the frame of the JULIUS project. The main driver in the design and implementation of the SimServer has been the parallel computational efficiency for transient aerodynamic and aeroelastic simulations. The system developed in JULIUS is built around the unsteady Euler solver AIRPLANE+, the Finite Element structural analysis code LAGRANGE and a spatial coupling library based on the neutral surface approach. These components are shortly described in the next sections. Both static and transient dynamic aeroelastic problems can be solved. The elastic response of the structural model can be computed either by direct access to the flexibility matrix – pre-computed by a LAGRANGE calculation – or by the modal approach. In the time-dependent aeroelastic case the single disciplines are strongly coupled via an implicit approach with sub-cycles. For the time integration an implicit Jameson type two step backward Euler scheme is used in both disciplines. Additionally implemented are important analysis operations like the calculation of aerodynamic loads, the extraction of arbitrary submeshes for visualisation and the calculation of several derived variables like total pressure loss, and others.

C++ has been chosen because of the downward compatibility to C which enables the tight integration of already existent and validated software packages written in other languages (Fortran77/90, C, C++). In the SimServer-system the EADS M in-house codes Airplane+ (parallel CFD-solver), the spatial coupling routines LFCL and a CSD modal solver are integrated via C-, Fortran77/90- and C++-libraries.

Parallelism is incorporated into the system by use of the message passing programming model. The use of the MPI-standard ([6], [7]) enables the parallel execution of aerodynamic and aeroelastic calculations transparently on both distributed and shared memory parallel platforms (e.g. MPP, WS- Clusters).

From the users' point of view the parallelism is hidden as much as possible. No separate parallel pre- and post processing has to be done, only at the beginning or at a restart the user has to specify the total number of processes on which the simulation should be performed. It should be noted that for a restart the number of processes is completely independent from the previous run.

General Design

In this section the general design is declared in combination with the steps necessary to perform a simulation. At the top of the hierarchy (see figure below) the simulation director resides who initialises the common components like the logical model and the motion controller.

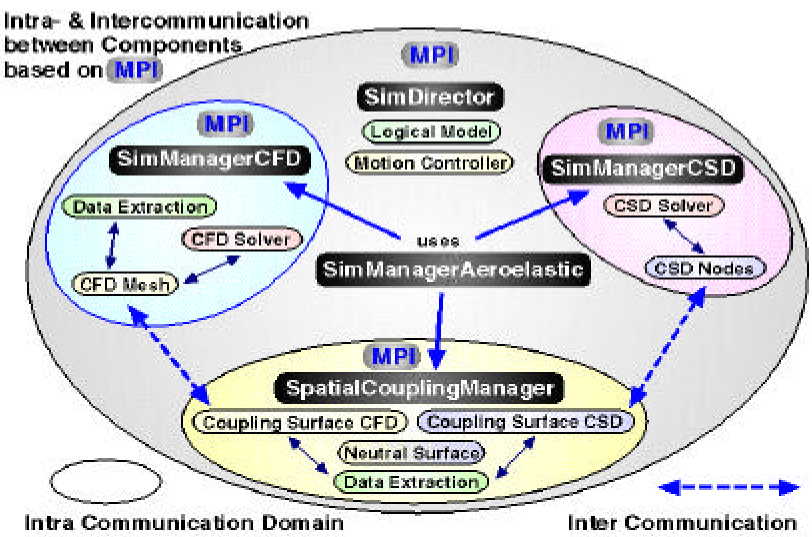


Figure 1 – General Layout of the JULIUS SimServer

L. Fornasier

The simulation director instantiates depending on the simulation type the corresponding manager and delegates the control to him. In case of an aeroelastic calculation the aeroelastic simulation manager takes over the control and begins his own initialisation. First the initial communication domain is split up in three groups for CFD, CSD and the spatial coupling. In each domain the corresponding managers are called for the discipline specific initialisation. The CFD manager first initialises in his domain the CFD volume mesh. This parallel MeshLib object - a parallel OO mesh management library developed by the third author - reads in from a file the volume mesh in parallel and takes care about the on-the- fly partitioning. After that the spatial coupling surface grid is extracted in parallel and sent to the spatial coupling domain over an inter-communicator. Then the CFD solver is called to perform the necessary preprocessing in parallel like the creation of the dual grid, the multigrid agglomeration, etc. based on the volume mesh currently loaded. It is worth notice that the solver uses the same large data arrays in memory as stored in the mesh object. No duplicates are necessary which ensures memory efficiency. At the same time the spatial coupling manager initialises himself. First the CSD surface grid together with the neutral surface geometry definition are read in. Then over the inter-communicator to the CFD domain the spatial coupling surface is received. At the moment the spatial coupling domain consists of only one process due to the sequential spatial coupling method LFCL. Parallel to the CFD solver initialisation the LFCL-object is initialised whereby the necessary mappings between the different surface meshes are computed or only updated in case of a restart. In the CSD domain the CSD solver has to be initialised with the modal forms of the structural model. In the transient case only a sequential modal CSD solver is currently integrated, for static cases also a parallel sparse matrix solver based on the stiffness matrix can be used.

After the initialisation is completed the aeroelastic simulation manager starts the static or transient simulation. For a transient manoeuvre simulation an implicit algorithm with sub-cycles is executed by the manager to drive both disciplines strongly coupled and consistently over the time.

The basic components of the aeroelastic system

The computational model consists of three main building blocks, the aerodynamic code, the structural analysis code and the fluid-structure coupling module. They are shortly described in the following.

The CFD Solver AIRPLANE+

The flow solver used here is an in-house development based on Jameson's AIRPLANE unstructured code [9]. The EADS code, dubbed Airplane+ , solves the governing equations, Euler or RANS, on unstructured meshes composed of tetrahedra, pyramids, prisms and hexahedra by means of a Finite Volume method on the dual grid. An efficient edge-based data structure has been used for which the underlying primary grid is completely transparent. Convergence to steady state is accelerated using the agglomeration multigrid algorithm. In contrast to its structured counterpart, where coarse grids are obtained by deleting every other grid line, the creation of coarse grids is not so straightforward for unstructured grids. A possible way to achieve this goal is agglomeration [10], where coarse grid control volume are obtained by fusing neighbours on the fine grid control volumes. The agglomeration procedure is applied recursively to obtain the desired sequence of coarser grids. To speed up the computation, this algorithm has been parallelised.

For the manoeuvre the ALE (Arbitrary Lagrangian Eulerian) formulation of the equations is used, such that manoeuvres can be simulated correctly. In this formulation both the object and the fluid can move and consequently edge velocities must be introduced. These are combined with the edge velocities obtained from the motion of the structure and computed in such a way that the Geometrical Conservation Law is satisfied. Due to the deformation of the structure, also the volume grid must be deformed, such that during each iteration the mesh is valid, i.e. no negative volumes are present. The relative displacements compared to the previous iteration are the solution of a diffusion equation, where the diffusion coefficient is inversely proportional to the volume of the primary grid cell. A standard Galerkin Finite Element discretisation is used and solved with GMRES, in which many of the parallel data structures of the flow solver can be used. Although it cannot be guaranteed that no negative volumes occur, the algorithm proved to be quite stable for not too complex geometries.

An unsteady problem is simulated and consequently the unsteady equations must be solved. As the explicit time step restriction for stability is much smaller than the physical time scales involved in the computation, the following unconditionally stable implicit time integration method is used:

$$\frac{1}{\Delta t}\left(\frac{3}{2}U^{n} - 2U^{n-1} + \frac{1}{2}U^{n-2}\right) = \mathrm{RES}\left(U^{n}\right). \qquad (1)$$

Equation (1) is used for both the CSD and the CFD solver. For the former this leads to a linear set of algebraic equations for the state U^n, which can easily be solved. For the CFD solver however, equation (1) is a non-linear set of equations for the state U^n, which must be solved iteratively. The dual time stepping approach [9] is used, in which a set of pseudo-time derivatives are added to the left hand side and equation (1) is considered as the steady (four dimensional) state of the new set of equations and consequently the multigrid acceleration technique can be still used, albeit slightly modified.

The Structural Analysis Code MBB-Lagrange

The structural analysis and optimisation package Lagrange is an EADS-M in-house development designed primarily for optimisation tasks [11]. The package allows optimisation of Finite Element structural models including treatment of buckling static (stress and displacement), dynamics (both in time and frequency domain), static and dynamic aeroelastic and calculation of analytical sensitivities in consideration of manufacturing restrictions. Design variables can be of type 'sizing' (element thickness) or 'shape' (element geometry), and a suite of gradient-based optimisation algorithms is available. The structure is modelled by finite elements (FE) similar to those used in NASTRAN assuming geometrical and material property linearity. For dynamic applications, a reduction of the elastomechanical model in a modal basis consisting of a series of natural eigenmodes and diagonalised mass and stiffness matrices can be computed.

The spatial coupling aeroelastic interface library LFCL

In consideration of the non-mapping characteristics of the CFD and CSD grids normally used in the two different disciplines, the 'neutral surface' approach has been selected for the implementation of the spatial coupling algorithms. In this approach an intermediate layer between the CFD grid and the CSD Model is defined, onto which the CFD and CSD nodes involved in the data exchange are mapped and the aerodynamic loads and the elastic deformations fitted by appropriate analytical functions. In the present implementation the NURBS patches of the CAD model are used as intermediate layer (neutral surfaces). Schematically the set-up of the coupling interface is done as follows:

1. Identify the CFD nodes and the CSD elements involved - CFD: all surface nodes belonging to the aircraft wetted surface; CSD elements: two dimensional elements (e.g. triangles, quads) formed connecting the structural nodes accepting external forces - ;
2. Project all the CFD and CSD nodes of step 1 onto the CAD patches and compute the physical (x,y,z) and parametric variables (u,v) of the projected point
3. Using a least-square approximation build a non-uniform B-spline fitting (B-splines) each set of CFD and CSD nodes lying on one surface patch
4. Using the actual values of the physical coordinates and pressure at the CFD nodes, integrate the aerodynamic load acting on each CSD element by performing a Gauss-quadrature using the CFD fitting of step 3 in the parametric space u,v
5. Compute the equivalent nodal 'local' forces at the CSD element's nodes by using the equilibrium of total forces and moments; the CSD nodes are supposed to be 'moment-free'
6. Sum up all the 'local' nodal forces contributions coming from all elements sharing a node: the set of these 'global' nodal forces is passed to the CSD analysis code;
7. Using the actual values of the elastic deformations at the CSD nodes, interpolate their value at the projection of the CFD nodes using the fitting of step 3 in parametric space;
8. Correct the deformations taking into account the vector distance between the actual position of the CFD node and the corresponding CSD element: the set of the corrected values of the interpolated deformations at the CFD surface nodes are passed to the mesh deformation algorithm (alternatively, the elastic deformations at the NURBS control point can be computed and the geometry of the deformed patch redefined).

Due to the iterative character of the aeroelastic coupling problem, advantage has been taken from the fact that – assuming a given shape for the fitting functions – the least-square problem becomes linear with respect to the fitting base. Hence, steps 2 and 3 – the most time-consuming parts of the whole sequence - can performed only once, provided that no mesh refinement has taken place between two successive iterations.

In practice, more complicated algorithms has been implemented for taking into account efficiency and robustness considerations and fulfilling requirements stemming from the necessity to extend the fitting algorithms to the treatment of trimmed patches and missing parts in the CSD modelling. The final implementation has been done in form of a C/C++ compatible FORTRAN90 library (LFCL).

Time Accurate Simulation of a 5g-pull-up Manoeuvre for a generic X-31-Experimental Aircraft Configuration

The main objective of the JULIUS project was the development of 6S, a Problem Solving Environment, designed to provide an integration platform for software modules required to simulate complex, large scale, industrial problems. As part of the industrial demonstrator suite defined to assess 6S capabilities when performing large scale, multi-physics computational simulation, EADS-M has simulated the fluid-structure interaction during rapid aircraft flight manoeuvres at transonic conditions by use of an advanced, time-accurate CFD-CSM coupling methodology. The intermediate technology demonstrator suite – used to steer, monitor and validate the software developments during the JULIUS lifetime – used the X-31 Aircraft Configuration as reference for building up a series of test cases of increasing complexity with respect to the requirements to the JULIUS technology components.

The final demonstrator selected by the EADS JULIUS Team deals with the numerical simulation of aeroelastic phenomena that occur during sustained or instantaneous flight manoeuvres. It is centred around the tightly coupling of fluid dynamics and structural analysis simulation methods for the solution of a transient aeroelastic problem in a parallel computing environment – the SimServer-system. The simulated manoeuvre can be considered the initial segment of a more complex flight manoeuvre named after the late Dr. Herbst – Leader of the MBB X-31 Design Team – 'Herbst Manoeuvre' which – exploiting the X-31 post-stall capability – allows the aircraft to reverse the flight direction in a very quick time by performing a 180 degrees roll about the instantaneous velocity vector (see Figure 2 below: the simulated segment of the manoeuvre lies inside the yellow line).

The following components have been involved in the set up and the execution of the simulation task:

- The 6S CAD Repair tool for importing, uncluttering and healing the X-31 CAD Model and for generating the NURBS description of the filtered model into the MEMCOM data structure; as the NURBS-based grid generator became operational in the late stages of the project, the (alternative) direct link between the CADR module and the mesh generator suite using the standard FLITE file format has been used
- The FLITE/DELTA grid generator suite for generating the volume mesh around the configuration;
- The AIRPLANE+ in its ALE-Euler-mode code for performing the flow analysis;
- The in-house structural analysis LAGRANGE code for performing the structural analysis at each time step; for sake of efficiency the structural eigenmodes computed by LAGRANGE have been used instead of the equivalent structural stiffness matrices;
- The LFCL library (CSD-CFD interface) for transferring the data between the CFD and the CSD modules aeroelastic coupling;
- The E2D (Extraction of Engineering Data) for the extraction and processing of the design-relevant data from the aerodynamic and the structural data;
- Visualisation tools like Vipar, TECPLOT

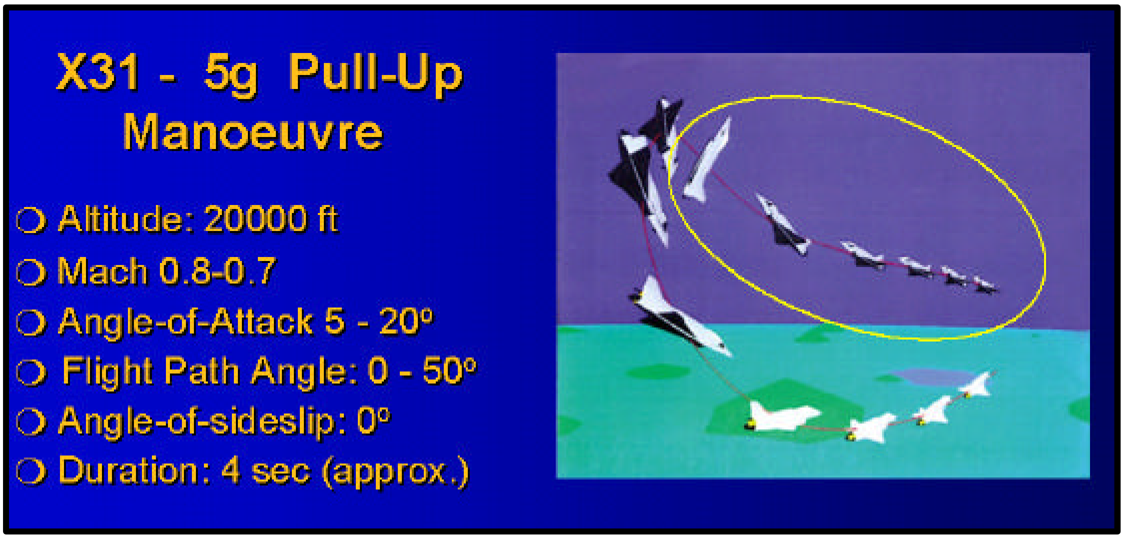


Figure 2 - Flight parameters of the simulated 5g –pull up manoeuvre

Set Up of the Aerodynamic and Structural Models

Starting point for the demonstrators was the availability of a detailed description of the X-31 Aircraft Configuration in form of a CAD model and of a FEM Structural Model of the complete configuration used during the Preliminary Design Phase of the X-31 Aircraft development and a more detailed Structural Model of the wing from the Definition Phase.

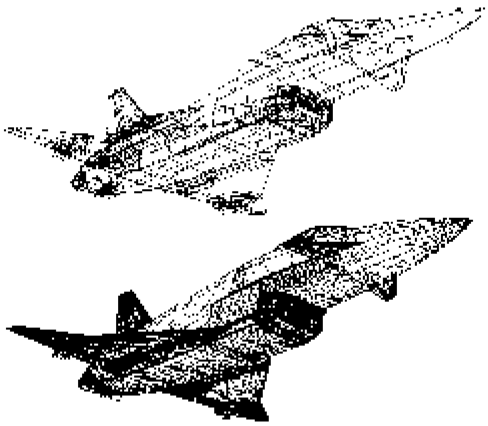


Figure 4 – Generic X-31– CAD Model – Visualisation of IGES Surface

The full representation of the X-31 CAD Model in IGES entities consists of more than 400 NURBS patches – both trimmed and untrimmed – and has a size of about 15MB. As the CAD Model was defined for constructive purposes, it contains quite a lot of geometrical and topological imperfections – e.g. gaps, overlaps, missing parts – that implies a large amount of 'CAD-Repair' for obtaining a waterproof surface as required by the mesh generators. Due to the complexity of the geometry and the size of the model, this test case resulted in a challenging benchmark for evaluating the capability of the JULIUS-CADR tool, a highly automatic, interactive procedure for obtaining 'waterproof' surface models from CAD geometries imported via IGES format.

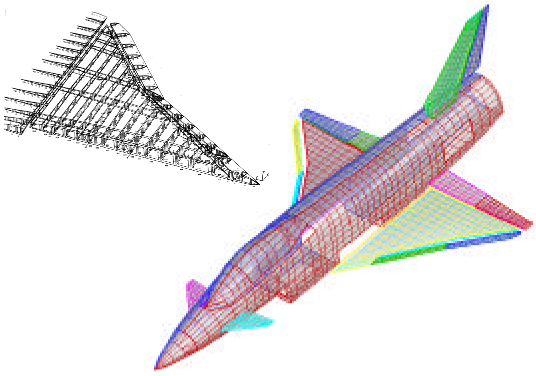


Figure 3 – Generic X-31 – FEM Structural Models

The FEM structural model contains approximately 25000 DoF (for half configuration). The 1660 FEM nodes, 'receivers' of the aerodynamic loads are assembled into 1460 quads and trias, grouped into 8 regions, each representing a configuration component – e.g. fuselage, wing slat, vertical fin. The relatively small size of the FEM structural models in comparison to the CFD surface grids and the different grid topologies used by the two disciplines set severe functional requirements on the fluid-structure spatial coupling algorithms. Of special interest is here the capability to extract a 'smooth' distribution of geometrical deformations at the CFD surface nodes from a rather sparse base of elastic deformations computed at the structural nodes. In addition, CFD grid deformations are to be provided also at configuration regions that have not been modelled in the FEM models – see Figure 5 - like engine air intake, undercarriage doors, airbrakes and vertical fin fairing.

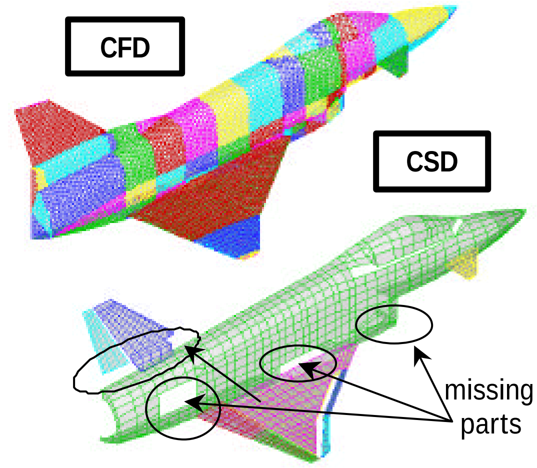


Figure 5 – Generic X-31 – Comparison of CFD and CSD surface meshes

The main steps involved in the demonstrator are as follows:

1. Import the CAD model into 6S
2. Repair the CAD model and generate the NURBS description of the model surface
3. Define and generate the Aeroelastic Neutral NURBS Interface
4. Generate the unstructured volume grid for the CFD analysis
5. Update (initialise) the aircraft position and velocity vector along the flight path
6. Compute the mesh movement
7. Compute the aerodynamic flow solution
8. Transfer the aerodynamic load onto the Neutral Interface
9. Compute the aerodynamic loads at the structural nodes from the Neutral Interface
10. Perform the structural analysis
11. Transfer the structural node deformations to the Neutral Interface
12. Update the surface NURBS parameters corresponding to the aeroelastic deformations
13. Deform/adapt the volume mesh according to the surface deformations
14. Iterate steps from 6 to 12 until convergence for the aeroelastic deformations is achieved
15. Extract the unsteady aerodynamic and structural data
16. Repeat the steps from 5 to 15 until the manoeuvre is completed

The SimServer system has been configured for the execution of the above sequence.

Simulation Results

After that the set up of the simulation problem, the preparation of the CFD and CSD models and a preliminary functional check of the SimServer environment have been done at the 16 processors -**Sgi** Origin 2000 computing facility at EADS-Ottobrunn, all the models together with the simulation software environment have been transferred via ftp to UWS, where the complete manoeuvre simulation run has been run on the 32-processors **Sgi** – Origin 2000 computer facility of the Civil Engineering Department of the University of Wales, Swansea (UWS). The CFD grid consisted of about 55,000 surface points, and 320,000 points in the volume mesh around the aircraft (corresponding to approximately 2 millions of tetrahedrons). The CSD model consisted of about 25,000 degrees of freedom. The aerodynamic loads have been mapped on 1460 surface elements, involving the calculation of 1660 equivalent nodal forces. The elastic deformations at the 1660 nodes have been transferred to the CFD surface grid via the mapping onto 320 neutral surfaces. On each neutral surface a maximum of 400 control points have been used to define the fitting B-spline functions. The dynamic coupling between the CFD and the CSD solver has been realised using the structural modal representation. In the *modal representation* of the structure the set of deflection distributions corresponding to the natural vibration modes of the structure is used to define the dynamic characteristic of the elasto-mechanical system. The eigenmodes form a base

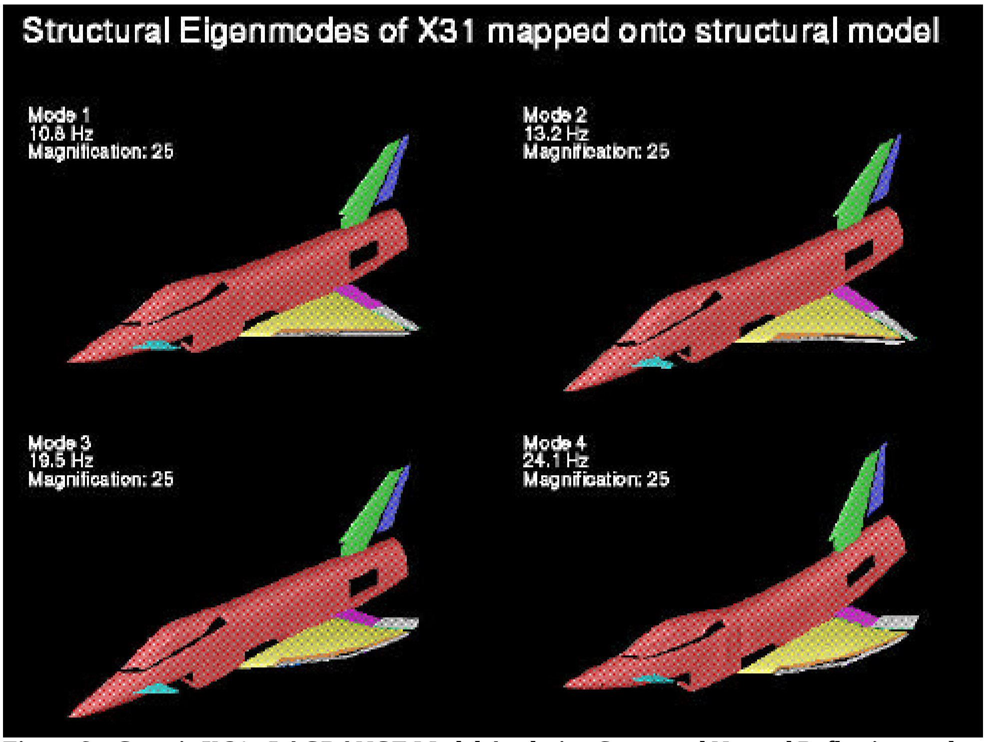


Figure 6 – Generic X-31 - LAGRANGE Modal Analysis – Structural Natural Deflection modes mapped onto the CFD aerodynamic model

in the mathematical sense for the motion or deflection of the elastic surfaces. These form an orthogonal basis (in a special metric defined by mass distribution). In the simulation of the pull-up manoeuvre, the first ten eigenmodes – i.e. the natural modes corresponding the ten lowest frequencies – have been used, as it is known from experience that dynamic behaviour is determined by the lowest frequencies. The corresponding structural deformations are shown in the Figures 6 and 7. In Figure.6 on the previous page, the deformations of the CSD model are represented; while in Figure, the mapping of the structural deformations on the CFD surface grid is shown. As can be seen, the spatial coupling algorithms developed in JULIUS allow to obtain a smooth distribution of the deformation even in the regions of the aircraft surface where the CSD model is not available – e.g. air intake, under-carriage doors, vertical fin fairing, airbrakes. The following pictures are the two initial frames from the animations built using the representations of the first four eigenmodes at different time intervals. The animations have been shown at the JULIUS Final Review.

The numerical simulation of the 4 seconds manoeuvre has been modelled using a time accurate coupling in time with a physical time step equivalent to 4 milliseconds. – Execution time of the 1000 steps has been approximately 35 hours wall clock (about 30% dedicated to I/O). During the simulation about 2 Gbytes of output results of aeroelastic data have been extracted and stored for later post-processing. For the JULIUS Final Project Review this output has been synthesised in a series of animations of the aeroelastic effects being computed during the demonstrator simulation of the full aircraft. The information being provided consists of the concurrent visualisation of time-dependent aerodynamic and structural field data. The final frames from these animations are presented in the next page of this paper, Figures 8 and 9.

In the first animation, the evolution in time of the surface pressure is shown on the aeroelastically deformed CFD grid. The second animation shows the evolution of the aeroelastic deformations as mapped on the CFD grid. For sake of visualisation, the structural deformations have been magnified. For reference, the evolution of the flight parameters, the instantaneous of lift and drag extracted by integration of the surface pressure, the weightings of the generalised co-ordinates corresponding to the first three natural deflection modes and the wing tip vertical displacement has been shown.

The initial oscillations shown by the force and deformation curves are due to the effect of the aerodynamic loads having been applied impulsively to the structure. Although this occurred unintentionally, the calculation has proven the robustness of the system with regard to the modelling of quite complex dynamic effects.

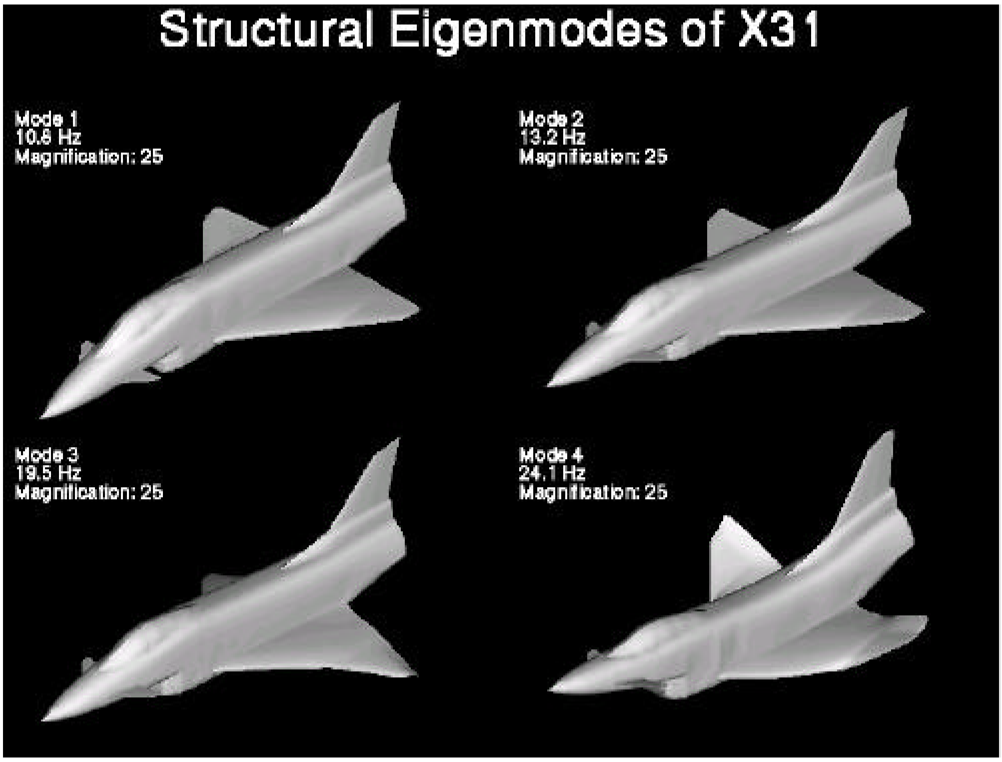


Figure 7 – Generic X-31— LAGRANGE Modal Analysis – Structural Natural Deflection modes mapped onto the CFD model

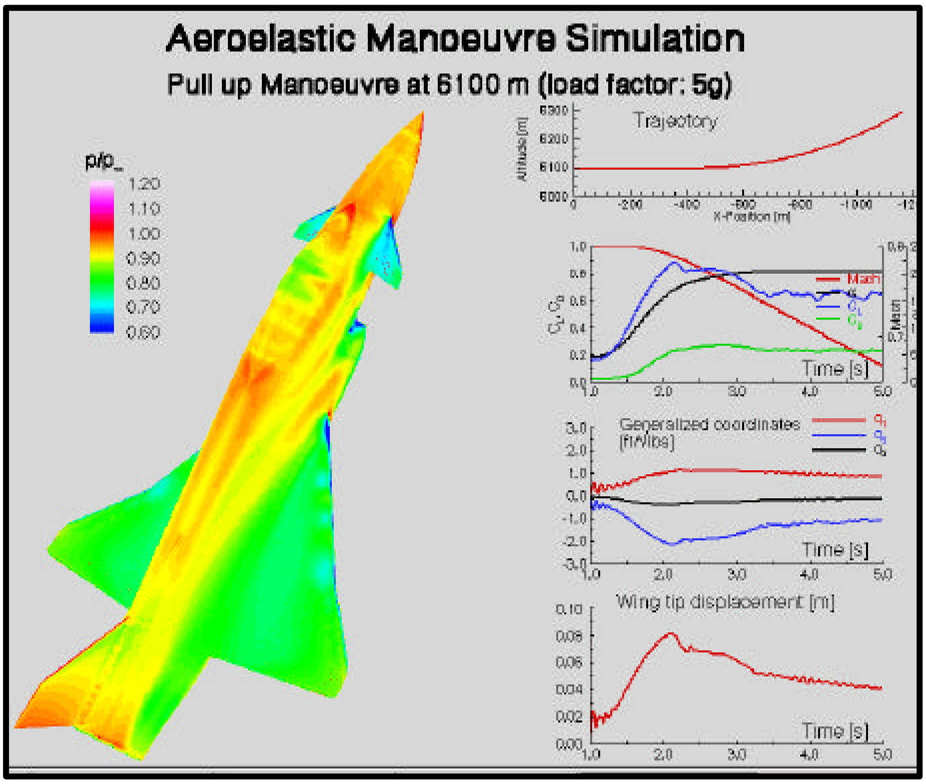


Figure 8 - Generic X-31– Animation of Instantaneous Surface Pressure (Frame at the end of the 5g –pull-up manoeuvre)

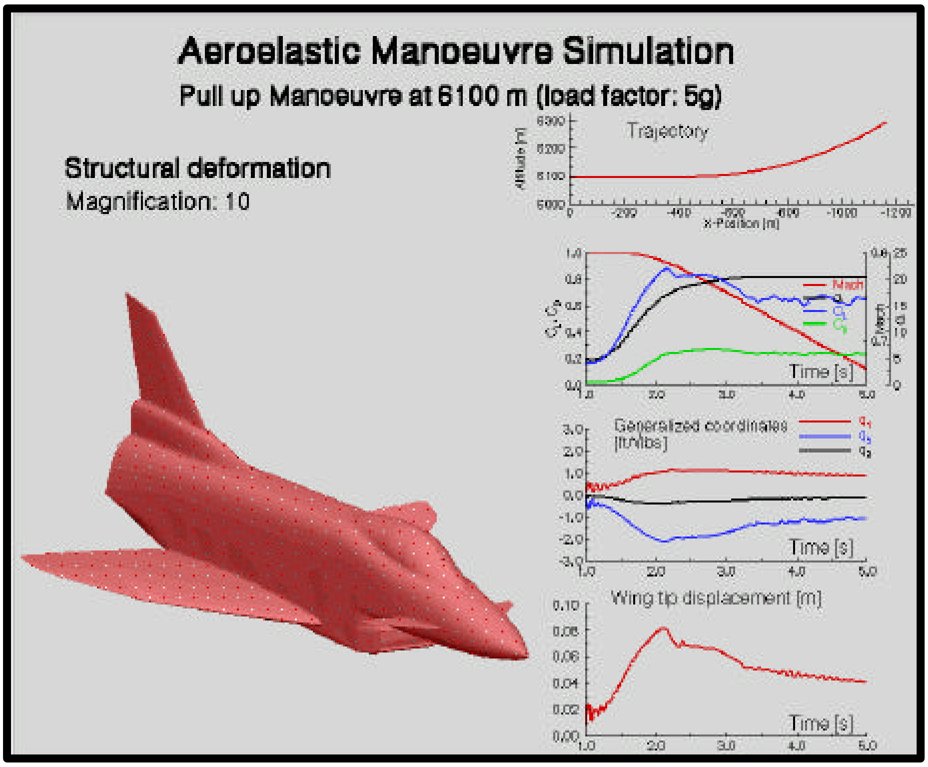


Figure 9 - GenericX-31 – Animation of Aeroelastic Deformations mapped onto CFD Grid (Frame at the end of the 5g –pull-up manoeuvre)

Conclusions

Aeroelastic phenomena occurring during sustained and instantaneous high-g flight manoeuvres can and do affect flight performance due to wing and control surface deformations. These aeroelastic effects should be assessed in the very early design stages in order to provide the potential customer with consolidated performance figures and avoid extensive and expensive ground and flight testing. Numerical simulation for this kind of problems requires coupling of non-linear fluid simulation and structural dynamics analysis methods. At the targeted flight conditions unsteady, separation-induced flow conditions are most likely to occur and therefore time-accurate methods have to be applied to capture the major time-averaged characteristics.

Within the framework of the JULIUS EU-Project, EADS-M with the assistance of other project partners has set up a pilot version of a numerical simulation environment suitable to compute the aerodynamic loads and performance of an aircraft flying rapid manoeuvres by taking directly into account the aeroelastic coupling problem, i.e. the unsteady flow-structure interactions. By an objected-oriented approach, an unstructured EULER CFD solver and an in-house structural dynamics analysis package have been wrapped into an aeroelastic simulation server that sets up and steers the numerical simulation process during the simulation of a flight manoeuvre in a very efficient and user-friendly way. The transformation of the aerodynamic loads into structural nodal forces and the conversion of elastic deformations computed at the structural nodes into the CFD surface grid is obtained by algorithms based upon NURBS fittings on suitable neutral surface interfaces. This approach allows aeroelastic couplings even in presence of very different CFD and CSD grid characteristics – e.g. element type and size, grid topology. For sake of consistency, a fully implicit algorithm is used for the coupling in time. The CFD mesh generation uses the unstructured grid technology developed at the University of Swansea. The mesh package features capability to generate meshes on parallel, and solution adaptive algorithms.

The functionality of the simulation system has been successfully demonstrated for the simulation of an assigned 5g-manoevre of the Generic X-31 at transonic speed. The simulation results, obtained running the multidisciplinary system on a 32 parallel processors platform for a total of about 35 hours wall-clock time have been presented. The paper has addressed the physical problem, the numerical and high performance computing requirements, and described the technical and computer science-related solutions adopted in the present approach.

The present application has to be considered as a preliminary, pilot version of a more complete, future simulation system that can be applied to the numerical simulation of aircraft manoeuvres as complementary tool to ground and flight testing.

Acknowledgements

The SimServer and the Final Demonstrator activities has been carried out within the frame of the ESPRIT Project EP25050 JULIUS co-ordinated by BaeSystems, Sowerby Research Centre. The calculations for the simulation has been done at the Computing Centre of the University of Wales, Department of Civil Engineering Department with the support of the staff of Prof. Nigel Weatherill. The FEM Model for the X-31 Aircraft has been prepared by Martin Stettner, Franz Weiss and Johannes Schweiger from the Structural Dynamics Department of EADS-M. Daniele Parisse and Friedbert Muenster contributed to the development of the coupling library LFCL.

References

[1] L.Hutsell, D. Schuster, J.Volk, J. Giesing, M.Love, *Evaluation of Computational Aeroelasticity Codes for Loads and Flutter*, AIAA Paper 2001-569, 2001.

[2] H. Balzert, *Lehrbuch der Software-Technik - Software-Management, Software-Qualitaetssicherung, Unternehmensmodellierung*, Spektrum Akademischer Verlag, 1998.

[3] H. Balzert, *Lehrbuch der Software-Technik - Software-Entwicklung*, Spektrum Akademischer Verlag,1998.

[4] E. Gamma, R. Helm, R. Johnson, and J. Vlissides, *Design Patterns: Elements of Reusable Object-Oriented Software*, Addison-Wesley, 1995.

[5] B. Stroustrup, *The C++ Programming Language*, Addison-Wesley, 3rd ed., 1997.

[6] M. S. et al., *MPI | The Complete Reference*, The MIT Press, 2nd ed., 1998. Vol. 1, *The MPI Core*.

[7] W. G. et al., *MPI | The Complete Reference*, The MIT Press, 1998. Vol. 2, *The MPI Extensions*.

[8] G. Dahlquist and A. Bjork. *Numerical Methods*. Englewood Cliffs, NJ Prentice-Hall, 1974.

[9] A.Jameson. *Time-Dependent Calculations Using Multigrid, with Applications to Unsteady Flows past Airfoils and Wings.*, AIAA paper 91-1596,1991.

[10] D. Mavriplis. *Three dimensional multigrid for the Euler equations*. AIAA-Journal, Vol 30:1753--1761, 1992.

[11] J. Schweiger, J. Krammer, H. Hoernlein, *Development and Application of the Integrated Structural design Tool LAGRANGE*, AIAA-96-4169, 1996.