New Approaches in Numerical Aeroelasticity Applied in Aerodynamic Optimization of Elastic Wing

Nové přístupy v numerické aeroelasticitě aplikované v aerodynamické optimalizaci elastického křídla

Zkrácená verze PhD Thesis

Obor: Konstrukční a procesní inženýrství
Školitel: doc. Ing. Jiří Hlinka, Ph.D.
          Ing. Martin Komárek, Ph.D.
Datum obhajoby:
Keywords

aeroelasticity, computational, static, equivalent structure, aerodynamics, optimization, elastic, wing, flutter, transonic

Klíčová slova

aeroelasticita, výpočetní, statický, podobná konstrukce, aerodynamika, optimalizace, elastický, křídlo, třepetný, flutter, transonický
## Contents

1 Introduction .................................................. 1
   1.1 Overview .................................................. 1
   1.2 The objective of the thesis ............................... 1
   1.3 The novelty of the thesis ................................. 2

2 Static aeroelastic computations .............................. 3
   2.1 Introduction ............................................... 3
   2.2 Fluid-Structure Interaction ............................... 3
      2.2.1 Principle of coupling ................................ 3
      2.2.2 Transformation methods .............................. 4
      2.2.3 Chosen coupling method .............................. 5
   2.3 Aerodynamics Modeling .................................... 6
      2.3.1 Governing Equations ................................ 6
      2.3.2 Flow Solver .......................................... 6
   2.4 Structural Modeling ....................................... 6
      2.4.1 Matlab Based Finite Element Solver ................. 6
   2.5 Mesh Deformation .......................................... 7
   2.6 Design of Computational Aeroelasticity Tool ........... 8
   2.7 Summary .................................................. 8

3 Equivalent Beam Model ....................................... 9
   3.1 Introduction ............................................... 9
   3.2 Inverse Design ............................................ 10
   3.3 Cases Descriptions ....................................... 10
   3.4 Results .................................................. 11
   3.5 Summary .................................................. 13

4 Static Aeroelasticity: Validation of the Computational Aeroelasticity Tool ............................................. 13
   4.1 Introduction ............................................... 13
   4.2 Test Cases ................................................ 14
      4.2.1 Model ................................................ 14
      4.2.2 Test Cases Summary ................................ 14
   4.3 Results .................................................. 15
      4.3.1 Cases B: Wing-only geometry, beam stick and wing-box structural models 15
      4.3.2 Case C: Wing-fuselage geometry, wing-box structural model, $M = 0.85$ 18
   4.4 Summary .................................................. 19

5 Aerodynamic Shape Optimization of Elastic Wing .......... 20
   5.1 Introduction ............................................... 20
   5.2 Principle of Aerodynamic Shape Optimization .......... 20
   5.3 Tools .................................................... 21
   5.4 Test Cases ................................................. 21
      5.4.1 Common description .................................. 21
      5.4.2 Initial design ........................................ 22
   5.5 Results .................................................. 22
   5.6 Summary .................................................. 24
6 Numerical study of benchmark supercritical wing at flutter condition

6.1 Introduction ........................................................................................................ 26
6.2 CFD-CSM solver ............................................................................................... 26
  6.2.1 Flow and structural solver ........................................................................ 26
  6.2.2 Coupling scheme ......................................................................................... 26
  6.2.3 Mesh deformation ....................................................................................... 26
6.3 Test Case ........................................................................................................... 27
  6.3.1 Experimental Setup .................................................................................. 27
  6.3.2 Computational setup .................................................................................. 27
6.4 Results ............................................................................................................... 27
  6.4.1 Estimate of flutter dynamic pressure ......................................................... 27
  6.4.2 Estimate of the flutter boundary ................................................................. 30
6.5 Summary .......................................................................................................... 30

7 Conclusions ........................................................................................................ 31

7.1 Outcome of the Thesis .................................................................................... 31
7.2 Conclusion ........................................................................................................ 31
7.3 Perspectives ...................................................................................................... 31

Author’s publications ............................................................................................ 38

Author’s Curriculum vitae .................................................................................... 39

Abstract .................................................................................................................. 40
1 Introduction

1.1 Overview

The aeroelasticity is a discipline studying interaction of fluid, elastic and inertial forces, thus connecting together the fields of aerodynamics, elasticity and dynamics. The phenomena occurring due to interaction of the mentioned forces result in the performance changing or structural damaging effects. Therefore, the aeroelasticity concerns the aircraft designers since the earliest years of the aviation. The experimental research in this field started during development of the first airplane. That time the Wright brothers investigated the effect of the wing warping, applied to their Flyer biplane to provide the roll control, and also adverse effect of a propeller blade torsional deformation on the trust. Perhaps the first experience with the engined aircraft failure due to aeroelastic phenomenon was the flight of Samuel P. Langley’s monoplane. His attempt of the first flight failed due to the wing twist off caused by insufficient wing torsional stiffness. The phenomenon is known as the torsional divergence. During following years, the airplane designers encountered other aeroelastic phenomena such as the elevator flutter caused by insufficient torsional stiffness of the fuselage and the tail combined with unsuitable design of the control surfaces drive. The return to the monoplane design and increased speed of aircrafts brought the need to solve other aeroelastic issues - aileron effectiveness loss or reversal, wing flutter or wing lift redistribution. With reaching the supersonic and hypersonic speed new aeroelastic phenomena connected with a shock wave oscillation occurred, such as control surface buzz or panel flutter [1].

In the early years of the aviation the solution of aeroelastic problems was sought by a trial and error. Thus, problems were solved in late stage of the design process and often led to accidents during flight tests. In the course of the aeronautical engineering development, the theoretical investigations and research were conducted to understand the aeroelastic phenomena. The theories solving unsteady aerodynamics were established by Wagner [2] and Küssner [3]. The developments in the unsteady aerodynamics allowed to create theories predicting the wing flutter in subsonic speeds by Küssner [4] and later by Theodorsen [5]. The Theodorsen’s work created a basis for the strip theory, which has been further developed [6]. Nowadays, commonly applied method for aerodynamic prediction in aeroelastic computations is based on the doublet-lattice method, [7, 8]. This essentially linear method is capable of relatively accurate prediction in the subsonic conditions with no flow separation but fails in transonic flow or flow with extensive separation. Therefore, it has been common practice to correct results of the linear aerodynamic methods by the wind tunnel measurements. The advances in the development of computational methods based on the finite volumes started an extensive research of the aeroelastic solvers employing CFD simulations potentially reducing the number of tunnel or flight tests. The research in this field probably started by Bendiksen [9], who has been followed by Lee-Rausch [10], Alonso [11], Thomson [12], Feng [13] and many others, focusing on both static and dynamic aeroelastic simulations. The recently established activity in NASA [14, 15] focusing on validation of tools for high-fidelity flutter predictions in transonic speeds highlights the effort given to the research in this field.

1.2 The objective of the thesis

The trend in aircraft design is focused on the increase of aircraft overall efficiency. One of the aspects is the fuel consumption decrease, aiming to reduce the operational costs and emission of greenhouse gases. This can be obviously realized by the optimization of a propulsion system or by design of a lighter airframes and aerodynamically more efficient shapes, e.g. long slender wing. The combination of last two mentioned leads to increased flexibility of the airframe and consequently to the change of the aerodynamic characteristics, stability margin or control surface efficiency. In addition, compared to stiffer wing, the aeroelastic effect can become stronger and more easily excited by rigid body motions or an input of a flight control system. The potential aeroelastic phenomena can occur in large range of speeds involving transonic regime, where the non-linear flow effects significantly influence the flutter
speed. Moreover, the complex flow around the wing, due to its shape and interaction with other components, such as nacelles, pylons, an engine flow and a fuselage, makes the aerodynamic design and analysis the challenging task.

All mentioned aspects ask for a simulation tool which is able to:

- cover full range of the flight envelope - increasing the design efficiency [16],
- include the aeroelastic effects in early design stage - minimizing the need for often costly redesign in later stages,
- resolve the non-linear aerodynamics - allowing the design optimization of the complex geometries.

The mentioned requirements disqualify common aeroelastic tools employing unsteady aerodynamic solvers based on linear methods such as a strip theory or a doublet-lattice method. The progress in Computational Fluid Dynamic (CFD) in the areas of numerical scheme stability, code effectiveness and turbulence modeling, created a reliable tool for unsteady non-linear aerodynamic predictions. Therefore, the CFD is the convenient method for application in high-fidelity aeroelastic simulation tool covering all mentioned requirements.

The thesis objective is to create a tool for simulation and design optimization of static aeroelastic models of the aircraft. The tool will be applicable in:

- computation of aerodynamic characteristics of an elastic airplane,
- aerodynamic design optimization of elastic wing (aircraft) giving the result closer to reality,
- conceptual design based on aero-structural optimization - design of aircraft shape together with airframe structure aiming for improvement of both aerodynamic and structural efficiencies.

1.3 The novelty of the thesis

The research in the field of aeroelastic simulation and design is important and concerns large variety of computational tools ranging from CFD capable to accept large deformation of boundaries, employing techniques such as mesh deformation [17, 18, 19, 20], chimera grids [21], immersed or embedded boundary conditions [22, 23, 24], through efficient and accurate time integration schemes of coupled fluid-structure equations [25, 26, 27], spatial coupling [28, 29] or recently the adjoint of coupled fluid-structure equations of high-fidelity solvers in multidisciplinary optimization [30, 31].

The thesis is developing the activity at Institute of Aerospace Engineering focused on aeroelastic simulation and design using modern techniques like the fluid-structure interaction and adjoint method [32].
2 Static aeroelastic computations

2.1 Introduction

The research in field of high fidelity aeroelastic computation has been conducted since past few years. The advances in Computational Fluid Dynamics, creating the standard tool for non-linear aerodynamic predictions, commenced the interest in coupling high fidelity flow solvers with the already matured Finite Element Method. The research focuses on wide range of computational methods involved in the aeroelastic simulations, as it was mentioned previously in the thesis, ranging from mesh deformation methods [17, 18, 19, 20] through effective spatial and temporal coupling of the essentially different domains [28, 29, 25, 26, 27].

In this chapter, methods involved in static aeroelastic computation are described. The emphasis is given to fluid-structure interaction methods with focus on the description of the basic principles and methods involved. The summary of the aerodynamic and structural modeling follows and is complemented by the information about the solvers applied in the thesis. The end of the chapter is dedicated to the implementation of the static aeroelastic simulation tool applicable for the analysis and design optimization.

2.2 Fluid-Structure Interaction

Two main approaches for fluid-structure interaction are distinguishable - monolithic and partitioned. The monolithic approach, often referred as the strongly coupled, combines the fluid and the structural state equations together and treats them as a single system of equations governing both problems guaranteeing a conservation of properties at the fluid-structure interface. The interaction between domains is treated synchronously. The advantage is the robustness of the approach. The implementation requires the special solver, thus the existing well-established fluid and structure solver cannot be used.

The partitioned formulation approach allows to combine the complex domains described by different approaches and can differ in size by order of magnitude. The difference in size of the systems is common in the real applications when usually a flow domain is much larger than a structural domain. The necessary information obtained by arbitrary flow and structural solvers is exchanged on the defined interface according to chosen coupling scheme which should satisfy several criteria - conservation of energy and loads, accuracy and efficiency.

2.2.1 Principle of coupling

The principle of the fluid-structure coupling is based on the conservation of the virtual work satisfying the conservation of energy. The virtual work performed by the aerodynamic load must be equal to the virtual work of the structural forces:

\[ \delta W = F_s^T \delta u_s = F_f^T \delta u_f \]  

(2.1)

where \( F_s \) is vector of forces acting on structural nodes, \( F_f \) is vector of forces at fluid nodes, \( u_s \) and \( u_f \) are structural and fluid nodes displacement vectors, respectively.

The fluid-structure coupling is often expressed by introduction of a coupling matrix \( H \) giving a relation between the displacement vectors of the fluid and the structure meshes:

\[ u_f = Hu_s \]  

(2.2)

The combination of equations 2.1 and 2.2 gives a relation:

\[ F_s = H^T F_f \]  

(2.3)

Thus, if matrix \( H \) satisfies the mentioned conservation criteria, it can be used for the transformation of the structural displacements to the fluid mesh and once it is transposed for the transformation of the aerodynamic load to the forces acting on the structural nodes.
2.2.2 Transformation methods

During the research on fluid-structure interaction, relatively large number of coupling methods has been defined and applied. Reviews of methods were published, among others, by Hounjet and Meijer [33], Smith et al. [34] and Boer [35]. Many articles focusing on the particular coupling method were also published by Cerbral and Löhner [36], Beckert [37], Zwaan and Prananta [38], Wendland [29], Rendall and Allen [28, 39] and others.

The nearest neighbor interpolation The nearest neighbor interpolation is a simple method of the information transfer from mesh S (structure) to mesh F (fluid). For the given point \( x_F \) in mesh F the closest point \( x_S \) in mesh S is found. Consequently, the value of variable in \( x_F \) is taken as the same as in point \( x_S \). Thus, the coupling matrix \( H \) is a Boolean matrix.

Weighted residual methods The initial assumption for the method is the conservation of displacements on the interface, which is given in continuous form:

\[
 u_s(x) = u_f(x) \text{ on } \Gamma
\]  

The approximate solution of the equation is derived using Galerkin method. It gives the transformation matrix in the form:

\[
 H = A_{ff}^{-1} A_{fs}
\]  

where \( A_{ff} = \sum_{j=1}^{n_f} \left[ \int_{\Gamma} N_f^k(x) N_f^j(x) dx \right] u_{fj} \) and \( A_{fs} = \sum_{i=1}^{n_s} \left[ \int_{\Gamma} N_f^k(x) N_s^i(x) dx \right] u_{si}, \) \( N_s \) and \( N_f \) are the basis functions for the structure and the flow.

The last step is the selection of the interface for integration of integrals for which a projection method must be employed, e.g. Gauss interpolation [36] or Intersection method [35].

![Figure 2.1: Principle of coupling by finite interpolation elements (source [37])](image)

Method of finite interpolation element The finite interpolation elements are special type of finite elements with no stiffness, mass or any material properties. They relate displacements (rotation) of the element node with those in any element point employing the shape functions. Usually, this kind of elements is used as a connection of compatible substructures in finite element analysis.

The principle of the coupling is illustrated in Figure 2.1. A node on the fluid grid is projected on the finite interpolation element. The space between the element and the fluid node is rigid, therefore the aerodynamic load is transformed in terms of the equilibrium of forces and moments. Those forces and moments can be interpolated to the structure nodes by using of shape function of the
finite interpolation element. Vice versa, the displacements and rotation of the structure nodes are interpolated back to the projected points by the same shape functions.

The relation between the displacement vector of the projected point \( u_\hat{s} \) and the displacement vector of the structural node \( u_s \) can be formulated as

\[
u_\hat{s} = \hat{H}_s u_s,
\]

the matrix \( \hat{H}_s \) contains \( n_s \) diagonal matrices \( H_{s_i} \) (the size is 6x6 in case of six degrees of freedom), which are composed of values resulting from the evaluation of the shape functions of the finite interpolation element. The relation between displacements of fluid node and its projection point is

\[
u_f = D_s u_\hat{s},
\]

the matrix \( D_s \) provides the rigid transformation of the displacements and rotations.

In order to formulate the transformation matrix \( H \), the equations 2.6 and 2.7 are combined together. Then

\[
H = D_s \hat{H}_s
\]

The projection of the fluid nodes onto interpolation element can be solved by the orthogonal projection.

Radial basis functions

Radial basis functions \([40], [19]\) are flexible and well-established tool for multivariate interpolation. The displacements at the fluid and the structure are approximated by an interpolant which has the form:

\[
s(x) = \sum_{j=1}^{N} \gamma_j \phi(\|x - x_j\|) + h(x)
\]

where \( \phi \) is a given basis function, coefficients \( \gamma_j \in \mathbb{R} \), the \( x_j \) are centers with known values (structural points), \( h(x) \) are first degree polynomials and \( \| \cdot \| \) denotes Euclidean norm. In many cases it is convenient to scale the basis function with a shape parameter \( \epsilon \), then the basic function is replaced by \( \phi_\epsilon(r) = \phi(\epsilon r) \).

In \([40]\), it is shown that the coupling matrix \( H \) is:

\[
H = A_{fs} C_{ss}^{-1}
\]

Both matrices at the right-hand side come from RBF approach. The square interpolation matrix \( C_{ss} \) of size \( N_s \times N_s \) (\( N_s \) is a number of structural nodes) consists, among others, of values \( \phi(\|x_{s_i} - x_{s_j}\|) \), while the radial basis function is evaluated only on structural nodes. The matrix \( A_{fs} \) (of size \( N_f \times N_s \)) depends on both fluid and structural nodes.

The interpolation by radial basis function is dependent on the radius of the support \( r \), which can be varied by the shape parameter. Other important choice is a choice of the radial basis function itself. According to \([35]\), the most robust, cost effective and accurate are: \( \epsilon \).

- Multi-quadric biharmonic splines
  \[
  \phi(r) = \sqrt{r^2 + a^2}
  \]

- Thin-plate spline
  \[
  \phi(r) = |r|^n \log |r|, \text{ even}
  \]

The parameter \( a \) controls the shape of the basis function.

2.2.3 Chosen coupling method

For further application in the thesis, the radial basis function method was chosen. The reasons are the accuracy and no need of complicated and computationally expensive search and projection algorithm as in case of the weighted residual or the finite interpolation elements methods.
2.3 Aerodynamics Modeling

The thesis objective is to create tool for high fidelity aeroelastic simulations and multidisciplinary design optimization. The tool should be applicable for flight speeds ranging over full flight envelope, potentially involving transonic regime or conditions with strong flow separation. Moreover, the aircraft design involves flow solution around complex geometries, such as wing-fuselage configuration with nacelles or pylons. Those requirements disqualify the linear aerodynamic prediction tools such as doublet-lattice method. Since the Computational Fluid Dynamic (CFD) solvers fulfill mentioned requirements for the flow solution, the CFD solver will be used for aerodynamic prediction in the aeroelastic computational tool.

2.3.1 Governing Equations

Viscous flow The flow of compressible viscous fluid is described by equations expressing the conservation laws - conservation of the fluid mass, conservation of the momentum and conservation of the energy. The derivation of the particular equations can be found in numerous textbooks of the fluid mechanics. Versteeg [41], focused on finite volume method, gave the Navier-Stokes equations governing the time-dependent three-dimensional fluid flow and heat transfer of the compressible viscous fluid.

Inviscid flow In certain cases, the viscous effect can be neglected. It is typical for aeroelastic applications when the flow is assumed fully attached with small perturbations [42]. Therefore, the viscous and thermal conductivity terms in Navier-Stokes equations are dropped and the system of equations reduces to the Euler equations governing the inviscid flow:

2.3.2 Flow Solver

The flow solver used in the thesis is CFD code Edge [43]. It is finite volume solver for unstructured grids which can solve 2D and 3D Euler and RANS equations, as well as the adjoint of the Euler and NS (frozen viscosity) equations [32]. The time integration uses the fourth order Runge-Kutta scheme. It employs local-time-stepping, local low-speed preconditioning, multi-grid and dual-time-stepping for steady-state and time-dependent problems. For the unsteady cases, the employed numerical scheme is a dual-time-stepping scheme[44]. The data structure of the code is edge-based. The solver can be run in parallel on a number of processors to efficiently solve large flow cases.

2.4 Structural Modeling

The simulation and prediction of aeroelastic effects require an appropriate structural model. The model should be able to describe behaviour of loaded structure in sufficient extent appropriate to its application. Slender structures, which are typical for airplane wings, can be represented by a beam stick model. The application of this model is also allowed by high chord-wise rigidity of the wing due to ribs. Usually, the beam is placed to the position of a wing elastic axis, which is a line of points where a bending loading does not produce torsional deformation and vice versa. Therefore, decoupling of bending and twisting is allowed and Euler-Bernoulli beam elements can be applied.

2.4.1 Matlab Based Finite Element Solver

In the frame of the thesis, the finite element solver was programmed in Matlab environment. It is linear elasticity solver working in two modes:

- Mode 1: Beam finite elements mode
- Mode 2: Prescribed stiffness and mass matrices mode
The purpose of this solver is to overcome cumbersome communication with commercial FEM packages via input files and to allow future development focused on direct communication between solvers. The important parts of finite element preprocessor and solver code in Matlab are given in Appendix ??.

The first mode can solve static deformation and modal analysis of the model consisting of beam elements (either Euler-Bernoulli or Timoshenko). Its advantage is capability of direct input of a model which is applicable for model parameterization purposes.

The second mode is capable of solving model consisting of arbitrary structural finite elements but model size is limited by size of memory Matlab can allocate.

**Solution of linear static elasticity by finite elements**  Solution of a linear static elastic problem using finite element formulation leads to system of linear algebraic equations in the form:

\[ Ku = F, \]  \hspace{1cm} (2.13)

where \( K \) is a stiffness matrix, \( u \) is a vector of nodal displacements and rotations and \( F \) is vector of corresponding nodal forces and moments.

**Implementation of inertial forces loading**  In both solver modes, the loading by inertial forces is implemented. Inertial forces are calculated according to Newton’s second law of motion with an assumption that only inertial forces act on a structure. Thus, the inertial forces can be calculated from the equation:

\[ F_i = M \ddot{u}, \]  \hspace{1cm} (2.14)

where \( M \), \( F_i \) and \( \ddot{u} \) are mass matrix, vector of nodal inertial forces and vector of nodal accelerations, respectively.

### 2.5 Mesh Deformation

The motion of deformable surface of an aeroelastic model must be captured by a fluid computational grid prior to calculation of new flow solution. Two methods of moving geometry treatment exist, remeshing and grid deformation. The first mentioned allows to capture arbitrarily large geometry deformation, but at high computational cost connected with recalculation of entire volume mesh. Moreover, risk of physical conservation loss exists due to possibility of large changes in the grid which may lead to reduced local computational accuracy.

Therefore, development of mesh deformation techniques, such as spring analogy, Laplace smoothing and radial basis functions interpolation, has began in recent years. Their advantage is conservation of mesh topology, i.e. number of elements, nodal connectivity and generally lower computational cost compared to the remeshing. Commonly, mesh deformation methods suffer by high risk of inverted elements occurrence as result of large geometry deformation.

**Spring analogy method**

The method, originally proposed by Batina [45], is based on treatment of computational mesh as a network of springs connecting mesh nodes. Deformation of the boundary is propagated to the volume mesh on the basis of static equilibrium between fictitious forces which are proportional to nodal displacements.

**Laplace smoothing method**

The mesh deformation method originate in Laplace smoothing which was originally employed for improvement of the computational grids [46]. Essentially, the propagation of boundary deformation
into the interior is based on iterative movement of the mesh nodes towards the center of the polygon (2D) or polyhedron (3D) created by adjacent nodes. The method is prone to produce inverted cells, therefore it often fails to propagate large boundary movement. The effort was spent in order to overcome this issue [47, 48].

Radial basis functions interpolation method

This method proposed in [49, 19] applies the similar idea to the one used in the fluid-structure interaction where the movement of the control points defined on the structure grid is interpolated on the CFD surface mesh. In application for CFD grid deformation, the control points are defined on the movable boundary of the fluid mesh. The boundary movement is interpolated into the fluid volume mesh. Compared to the spring analogy and Laplace methods, the computational cost is low, once the interpolation matrix is created. Method can handle large deformations and is applicable for structured, unstructured and hybrid meshes, because it is independent of the mesh connectivity.

2.6 Design of Computational Aeroelasticity Tool

The implementation of the computational aeroelasticity tool is based on the partitioned (coupled-field) formulation of the fluid-structure interaction. Therefore, it is possible to couple arbitrary separate flow and structure solvers independently of each other. Thus, the best suiting solver for particular domain and application can be employed. Moreover, the solvers can be separately improved and maintained to comply the state-of-the-art level in the specific field.

The basic task of the tool is sharing appropriate information between solvers on defined interface. The formulation of the interface employing Radial Basis Functions (RBF) ensures ability to couple independently discretized domains differing in size by several orders of magnitude. At the same time, the formulation satisfies the conservation of energy and loads and it is accurate and efficient [35].

Basic principle

The principle of the computational aeroelasticity process is following. The initial step is definition of the CFD and CSM models and an appropriate interface connecting the models. In both domains, the surface nodes are picked to define the interface. It is an obvious choice because aerodynamic loading acts through surface pressure. The interface is described by a coupling matrix $H$ which is constant during the aeroelastic computation. The main computational loop consists of sequential calls of the fluid and structure solvers and the relevant information transfer. In each iteration, the flow solution is calculated on the actual deformed shape and the forces on fluid surface mesh $F_f$ are transformed to the forces on structural nodes $F_s$.

Consequently, the forces are sent to the CSM solver and are applied on the initial structural model giving the vector of structural deformation $u_s$ due to actual aerodynamic loading. The displacements are interpolated to the moving boundary nodes in the fluid mesh.

The CFD surface mesh deformation is propagated to the volume mesh employing appropriate algorithm capable to solve rather large surface mesh deformation. In the extreme case, if mesh deformation fails, a remeshing technique can be employed. At the end of the iteration, residual values are calculated and compared with convergence criteria. If the criteria are satisfied, the loop will end and a static equilibrium state of the aeroelastic model will be obtained. Otherwise, the loop will be repeated.

2.7 Summary

Methods of definition of the fluid-structure interface, such as the nearest neighbour, weighted residuals, method of the finite interpolation elements and radial basis function, were presented in this chapter. The radial basis functions method has been chosen for the implementation in the computational
aerodynamics tool. The reasons for the choice of this method are accuracy, independence on the mesh connectivity and no need for computationally expensive search and projection algorithms.

The computational fluid dynamic method was applied for the aerodynamic predictions, because it can resolve non-linear features of the flow such as transonic shocks and the flow separation. Therefore, the tool is applicable for full range of a flight envelope of designed aircraft, which might improve the design efficiency.

The linear elastic solver was implemented in the Matlab environment. It is capable to predict deformation of the structure modelled by beam finite elements with direct input of beam stiffness parameters. The second option is to provide stiffness and mass matrices to the solver, thus matrices must be assembled using an external finite element preprocessor. Additionally, the influence of the inertial forces can be modelled by the solver. Inertial forces are calculated from the mass matrix according to Newton’s second law of motion. The reason for the implementation of own structural solver was to overcome complicated communication with commercial solvers. Moreover, the solver might be implemented in the computational aeroelasticity tool using direct communication via the random-access memory, in potential further development.

The computational aeroelasticity tool was designed to employ arbitrary flow, structural and mesh deformation solvers. It is based on the simpler principle of communication using I/O operations via hard-copied files. The influence of various settings of the flow solver on the convergence of an aeroelastic solution was tested. The results suggest that optimal settings are case dependent.

3 Equivalent Beam Model

3.1 Introduction

Simplified structural model still possesses its place in common aerospace engineering practice. It is widely used in both static and dynamic aeroelastic analyses and during the design of load control system [42]. It can be also applied in multidisciplinary design optimization aiming for estimation of favorable aerodynamic shape and structural characteristics combination during an aircraft conceptual design [50].

Different approaches for design of an equivalent structural model, either to real structure or to higher fidelity finite element, were studied. The Dunn [51] performed a study focused on matching of natural frequencies and mode shape given by ground vibration tests. He employed genetic algorithm in order to determine optimal wing stiffness and mass distribution of the model to match the experimental data. Algorithm updated physical parameters such as bending stiffness and mass. The problem of the solution uniqueness was addressed in the work. Resulting process was based on model complexity variation (number of parameters), and at the end, the model giving a good representation of experiment with minimum number of parameters was taken as unique solution. Similar approach was adopted by Trivailo et al. [52]. They studied different approaches, i.e. matching either dynamic or static response, or both of them. Genetic algorithm and artificial neural network methods were compared and uncertainty of solution was evaluated.

Other study focused on design of accurate beam finite element model is presented by Elsayed et al. [53]. In their work, method of stiffness estimation to match static deflection of more complex model (wing-box) was described and compared with other common approaches, such as analytical approach or empirical estimation of stiffness distribution. Presented method is based on sequential application of unit load on given segment of complex model. Subsequently, the resultant segment deformation gives the stiffness of segment.

Relatively lot of effort was given to the development of model simplification methods based on Guyan reduction [54]. The method works with stiffness and mass matrices directly - it does not give physical properties of reduced model. Thus, the model would not be applicable in the multidisciplinary design optimization.

In this chapter, method of equivalent beam derivation is presented. The goal is to create a simplified structural model giving similar static aeroelastic deformation as more complex structure. The model
will be applied in the aerodynamic shape optimization of the elastic wing.

### 3.2 Inverse Design

The method purpose is to find simplified structural model, in this case a beam stick model, equivalent to higher fidelity model. Equivalence meant here is similarity in the static aeroelastic response of the coupled fluid-structure model. Therefore, deformation of given aerodynamic surface (e.g. a wing) under certain aerodynamic load must be, in the best case, same regardless of applied structural model. The problem can be formulated as fitting of the beam model stiffness parameters in order to get known deformation.

The inverse method principle is following. Initial step is a definition of aeroelastic models. The first model is a aerodynamic model coupled to a higher fidelity structural model $M_1$ - a wing-box model or a full finite element model of the wing structure. The model $M_2$ is the same aerodynamic model coupled to a simplified structural model - a beam stick model. The model geometry is defined at the beginning and remains unchanged during the process. Initial guess of structural stiffness distribution along beam must be provided (e.g. analytically obtained rough estimation or educated guess).

In the next step the same loading is defined for both aeroelastic models. The loading of the aeroelastic model $M_1$ gives deformed wing shape, the target shape $X^T$. Finally, the fitting of aeroelastic model $M_2$ begins by adjusting its stiffness until the loaded shape of the model $X^A$ matches $X^T$. The adjustment is done through a gradient-based optimization method minimizing an objective function in form:

$$
\min F = \frac{1}{N} \sum_{k=1}^{N} \sqrt{\sum_{j=1}^{3} \sum_{i=1}^{m} (X^A_{ij} - X^T_{ij})^2}, \quad (3.1)
$$

where $N$ is number of loading cases, $m$ is total number of surface nodes in the aerodynamic model and $1 \leq j \leq 3$ represents x, y and z directions.

### 3.3 Cases Descriptions

**Models** Two types of coupled fluid-structure (aeroelastic) models were defined. In both models, the wing of the Common Research Model was used as an aerodynamic model (for more details about model origin see section 4.2.1). The employed Euler CFD grid is presented in the Figure 3.1.

![Figure 3.1: CFD model - overview (left) and detail (right)](image)

The aeroelastic model employing the higher fidelity wing-box representation of the structure was considered as a target model. Specifically, the wing-box model labeled as v14 described in the section ?? was considered. In the other aeroelastic model, the structural model consisted of beam elements. Both structural models are shown in Figure ??.

In all cases, the coupling between CFD and CSM model was done by Thin Plate Spline type of radial basis function, when the wing surface nodes were coupled to outside nodes of wing-box model or to all beam nodes.

**Loading** The goal is, in both aeroelastic models, to obtain the same static wing deformation due to aerodynamic load, thus the aerodynamic forces acting on the rigid wing surface at given flight
condition (Mach number, altitude, angle of attack, ...) might be chosen as the loading (labeled as $Q_a$). This loading results mostly in the displacement of the wing surface in vertical ($z$) direction due to bending, which is influenced by the bending stiffness parameter $I_{zb}$. Thus, the loading might be insufficient to fit other parameters such as bending stiffness parameter $I_{yb}$ and torsional constant $I_{xz}$.

Therefore, other loading cases were considered:

- loading $Q_z$ - positive unit forces in $z$ direction in each wing surface node
- loading $Q_x$ - positive unit forces in $x$ direction in each wing surface node
- loading $Q_t$ - positive unit forces in $z$ direction in wing surface nodes ahead of considered elastic axis and negative unit forces in $z$ direction in wing surface nodes aft of the elastic axis

The considered coordinate system is shown in Figure 3.2.

**Parameterization** The optimization based method requires suitable parameterization in order to find characteristics of equivalent structural model. Two types of parameterizations were tested. The first parameterization uses structural characteristics directly as parameters. Thus number of optimization variables grows with increasing number of beam elements. Physical feasibility of structural characteristic was imposed by bounds on parameters.

The second one uses constants of n-th degree polynomial as parameters, therefore number of variables depends on polynomial degree. Constraint functions and bounds on parameters were imposed in order to define space of feasible structural characteristic, which is close to reality such as:

1. negative gradients of polynomial functions - decreasing stiffness from root to tip
2. continuity of given characteristic at kink position
3. positive values of polynomial functions - strictly positive stiffness

Additionally, influence of beam stick model discretization was also tested. The test cases descriptions are given in Table ??.

Analytically calculated stiffness distribution along beam was the initial guess for minimization problem. In cases C-H the linear polynomial constants were fitted to initial beam stiffness.

The beam stick model and the wing surface mesh were placed into coordinates $xyz$. Each beam element is placed into local coordinate system $x_b y_b z_b$, thus its cross-sectional characteristics are related to this coordinate system.

![Figure 3.2: The coordinates system](image)

### 3.4 Results

**Multiple loading cases**

The inclusion of the three different load cases aims to solve the indeterminacy of the resultant values of the stiffness parameters $I_{yb}$ and $I_{xz}$. Only the cases employing linear polynomial parameterization were considered. Direct parameterization and quadratic polynomial parameterization cases were not tested. The reason is that the first mentioned, besides their tremendous computational cost, provide essentially unrealistic results. The second parameterization cases did not provided any considerable improvement of the results and the cost was higher compared to cases employing linear polynomial parameterization.
Additionally, the effect of different initial guess of the design parameters was tested. The case F2 was taken as a reference. In that cases (labeled as F3 and F4), the initial stiffness parameters were constant along span and the values were equal to the ones of the initial beam at the kink position.

The Table 1 shows the values of the objective function and the computational cost.

The resultant objective function values in the cases F3 and F4 with different initial guess are nearly the same as in the reference case F2. The computational cost is comparable as well.

<table>
<thead>
<tr>
<th>Case</th>
<th>$F_{min}$</th>
<th>$N_{F_{eval}}$</th>
<th>$N_{gr_{eval}}$</th>
<th>Total $N_{F_{eval}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>C2</td>
<td>6.8967</td>
<td>31</td>
<td>22</td>
<td>119</td>
</tr>
<tr>
<td>D2</td>
<td>4.8647</td>
<td>80</td>
<td>59</td>
<td>552</td>
</tr>
<tr>
<td>E2</td>
<td>1.6803</td>
<td>189</td>
<td>95</td>
<td>1329</td>
</tr>
<tr>
<td>F2</td>
<td>1.6831</td>
<td>400</td>
<td>72</td>
<td>1264</td>
</tr>
<tr>
<td>F3</td>
<td>1.6729</td>
<td>176</td>
<td>53</td>
<td>812</td>
</tr>
<tr>
<td>F4</td>
<td>1.7228</td>
<td>182</td>
<td>65</td>
<td>962</td>
</tr>
</tbody>
</table>

\[ a \text{ gradients were calculated by finite differences,} \]
\[ b \text{ calculation cost in terms of CPU time is proportional to number of optimization variables} \]
\[ \text{total number of objective function evaluation calls - objective values + gradients calculation} \]

Table 1: Results of multiple loading cases - the objective function and the computational cost

The resultant design parameters values are plotted in Figure 3.3. The plots suggest that the final bending stiffness parameter $I_{z_b}$ is independent on the setting of the inverse design method. Although the certain improvement is observable, the other two parameters are not still clearly defined.

The different initial guess of the design parameters led to same resulting values of $I_{z_b}$ and $I_{x_b}$ but results of $I_{y_b}$ were slightly different.

![Graphs showing structural characteristics of resultant beam stick models - multiple loading polynomial parameterization cases](image-url)
3.5 Summary

The method for design of simplified structural model was presented. The reference structural model was a wing-box model of a complex transonic wing with a kink and high sweep angle. The method is based on the minimization of an objective function which compares loaded wing shapes, the reference one with the wing shape of aeroelastic model employing current design of the beam structural model.

Different parameterization approaches were tested. The first one, used directly the stiffness parameters at each finite element as design parameters. Results show that the resulting distribution of the stiffness is irregular and does not agree with the distribution expected in reality. Moreover, the computational cost is tremendous. Therefore, the parameterization using polynomials in order to define the design values of the stiffness parameters was employed. The results suggest that linear polynomials might be sufficient to define the beam stiffness characteristics. The initial guess of the stiffness parameters was rough analytical estimation of the relevant characteristic.

Initial test employed single loading by aerodynamic forces but it was shown that it is not enough in order to clearly define all stiffness characteristics. Thus, other cases were tested employing three loads by unit forces in different directions. The results of the method were improved.

Further, the influence of different initial guess was tested. The case, which gave the best results in terms of the objective value, the computational cost and the resulting shape of the loaded wing, was taken as a reference. The results of all observed characteristics agreed with reference ones, except the parameter defining the wing in-plane bending stiffness.

The results suggest that the proposed inverse method might be applicable for design of simplified structural model of a complex wing geometry. There is still space for further development of the method, such as extension for design of beam stick model dynamically equivalent to higher fidelity structural model. In that case, an objective could be to minimize differences between natural frequencies of structures and mode shapes interpolated to a wing surface.

4 Static Aeroelasticity: Validation of the Computational Aeroelasticity Tool

4.1 Introduction

The application of the Computational aeroelasticity tool is presented in the chapter. Several cases have been performed differing in applied structural and aerodynamic models. The main focus of the chapter is the validation of the tool functionality.

Two different structural models were applied - wing-box and beam stick models. The beam model is equivalent to the wing-box model in the sense of static deformation. The model was designed according to the inverse design method described in Chapter 3. The cases employing the beam stick model were focused on validation of the inverse design method. Other aim was the evaluation of the simplified structural model applicability for static aeroelastic simulation and aerodynamic design optimization of the flexible wing. In the cases employing the wing-box model, the airframe weight was considered. The comparison of the wing-box and beam structural cases was done without weight consideration.

The applied aerodynamic models are transonic wing-only and wing-fuselage geometries. In the first case the flexible wing is considered. The other case, using a rigid fuselage combined with a flexible wing, evaluates the tool ability to handle more complex conditions. The considered flow models were compressible inviscid and viscous solved by Euler and Reynolds averaged Navier-Stokes (RANS) equations, respectively. In attached flow conditions, the less expensive Euler flow solution can give reliable estimation of the pressure distribution needed for the aeroelastic simulations. Therefore, the aeroelastic solutions employing the Euler flow and RANS flow were compared in order to evaluate the differences.
4.2 Test Cases

4.2.1 Model

The model of a common transonic transport aircraft, namely the NASA Common Research Model (CRM), has been applied in the tests. The model was originally intended for CFD validation studies [55], but it became standard model for other applications including aerodynamic shape optimization [56], aero-structural optimization [57] and aeroelastic tailoring [58].

The finite element models of CRM wing structure are provided on the website [59]. The coarse model of the wing-box labeled as v14 was chosen for the evaluation of the computational aeroelasticity tool. It consists of 4622 nodes connected to 8502 quad or tri shell elements (see Figure 4.1). The wing main structure mass distribution was calculated from the finite element model. The masses of the engines, nacelles, control surfaces, flaps and the fuel were not taken into account.

A beam stick model of wing structure was designed according to method presented in the Section 3. The model is equivalent to the wing-box model meaning the static aeroelastic deformation of the wing is comparable in given operating conditions. The wing structure weight is neglected. The beam stick model (Figure 4.1) consists of 22 nodes connected together by beam elements.

Figure 4.1: Wingbox (left) and beam stick (right) finite element model of CRM wing structure applied in test cases

The wing-only unstructured Euler mesh consists of 842837 nodes and about 4.5 millions tetrahedral elements, whereas hybrid unstructured RANS grid consist of 2644786 nodes and about 7.6 millions tetrahedral and prismatic elements. In case of the wing-fuselage geometry, the Euler and RANS meshes consist of 865982 nodes (4.7 millions elements) and 2831524 nodes (8.5 millions elements), respectively.

The fluid-structure interface was defined using radial basis function (RBF), particularly by Thin Plate Spline (TPS) function. The TPS is one of the most robust, cost effective and accurate RBFs for the fluid structure interaction (see section 2.2.2). The interface was defined between the wing surface nodes in aerodynamic grid and either the surface nodes of the wing-box structural model or all nodes in the beam finite element model.

The method applied for deformation of aerodynamic computational grid in test of the aeroelastic tool was RBF mesh deformation which is part of the CFD code Edge.

4.2.2 Test Cases Summary

The transonic flow condition listed in Table 2 were considered in majority of the test cases. The corresponding Reynolds number in case of RANS flow simulation is approximately $\text{Re} = 5184000\text{ m}^{-1}$.

The summary of the considered test cases is given in the Table 3. In the cases A and B, the lift coefficient required for the steady level flight at given operating conditions was prescribed, thus the angle of attack was depended on the particular flow simulation (Euler or RANS) and it is not given in the table. In the same cases, the weight of the wing main structure was included in the static aeroelastic
Table 2: Flow conditions

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cruise Mach number</td>
<td>0.85</td>
</tr>
<tr>
<td>Cruise altitude</td>
<td>11000 m</td>
</tr>
<tr>
<td>Static pressure</td>
<td>22632 Pa</td>
</tr>
<tr>
<td>Temperature</td>
<td>216.65 K</td>
</tr>
<tr>
<td>Dynamic pressure</td>
<td>11450 Pa</td>
</tr>
<tr>
<td>Air density</td>
<td>0.364 kg.m$^{-3}$</td>
</tr>
</tbody>
</table>

The masses of other components, such as engine, nacelles, control surfaces, flaps and fuel, were neglected, although it is obvious they might significantly influence the wing deformation and the aerodynamic load distribution at the static aeroelastic equilibrium. It was assumed that it is sufficient to include the wing primary structure masses in order to evaluate a capability of the static aeroelastic simulation tool to handle the weight loads. The inclusion of other masses is just a matter of a finite element model preparation, as long as other masses are modeled using finite elements or point masses, the tool is able to handle them.

Case Geometry Structural model Airframe weight $M$ $C_L$ $\alpha$
--- | --- | --- | --- | --- | --- |
A   | wing | wing-box | yes | 0.85 | 0.5 | - |
B1a | wing | wing-box | no  | 0.85 | 0.5 | - |
B1b | wing | beam stick | no  | 0.85 | 0.5 | - |
B2a | wing | wing-box | no  | 0.6  | -   | 5 |
B2b | wing | beam stick | no  | 0.6  | -   | 5 |
C   | wing-fuselage | wing-box | yes | 0.85 | 0.5 | - |

Table 3: Test cases summary

The intention of cases B was an evaluation of the tool capability to apply a beam finite element model in the aeroelastic simulation and compare the result with cases using the wing-box model. Since the applied beam model is equivalent to the wing-box model by the stiffness not the masses distribution, the weight was not included in the simulation.

In all cases, the both Euler and RANS flow simulations were considered and their influence on the static aeroelastic solution was evaluated.

4.3 Results

The section presents results of the most interesting test cases among ones listed in Table 3.

4.3.1 Cases B: Wing-only geometry, beam stick and wing-box structural models

The intention of the test cases was to validate an ability of the computational aeroelastic tool to handle a simplified structural model. The beam stick model was designed according to inverse design procedure described in the Chapter 3. Evaluation was done for both Euler and RANS simulations at two free stream conditions: $M = 0.85$, $C_L = 0.5$ and $M = 0.6$, $\alpha = 5^\circ$. The results of the static aeroelastic calculation were compared with the case using the wing-box model.

The computational cost and the resultant values of aerodynamic forces are given in the Tables 4 and 5. The results, in terms of the aerodynamic forces, of the static aeroelastic computation using the simplified structural model are in good agreement with results given by the simulation applying the wing-box model. The largest difference is in pitching moment given by the aeroelastic simulation.
### Table 4: Comparison of aeroelastic simulation results using wing-box and beam stick structural models - Euler and RANS flow at $M = 0.85$ and required $C_L = 0.5$

<table>
<thead>
<tr>
<th></th>
<th>Euler</th>
<th>RANS</th>
<th>Wing-box</th>
<th>Beam stick</th>
<th>Difference</th>
<th>Wing-box</th>
<th>Beam stick</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of coupling iter.</td>
<td>19</td>
<td>20</td>
<td>-</td>
<td>15</td>
<td>16</td>
<td>-</td>
<td></td>
<td></td>
</tr>
<tr>
<td>No. of CFD iter.</td>
<td>8183</td>
<td>8875</td>
<td>-</td>
<td>4931</td>
<td>4830</td>
<td>-</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$C_L$</td>
<td>0.4996</td>
<td>0.4929</td>
<td>1.3%</td>
<td>0.5008</td>
<td>0.4898</td>
<td>2.2%</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$C_D$</td>
<td>0.0126</td>
<td>0.0126</td>
<td>0.0%</td>
<td>0.0179</td>
<td>0.0175</td>
<td>2.2%</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$C_m^a$</td>
<td>-0.1080</td>
<td>-0.1125</td>
<td>4.2%</td>
<td>-0.0614</td>
<td>-0.0615</td>
<td>0.2%</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*a pitch moment is related to the quarter point of the wing mean aerodynamic chord

Using Euler flow solution at $M = 0.85$. In that case, the negative moment is larger about 4% in case of applied beam stick structural model. Other forces differences are up to 2.5%.

### Table 5: Comparison of aeroelastic simulation results using wing-box and beam stick structural models - Euler and RANS flow at $M = 0.6$ and $\alpha = 5^\circ$

<table>
<thead>
<tr>
<th></th>
<th>Euler</th>
<th>RANS</th>
<th>Wing-box</th>
<th>Beam stick</th>
<th>Difference</th>
<th>Wing-box</th>
<th>Beam stick</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of coupling iter.</td>
<td>7</td>
<td>8</td>
<td>-</td>
<td>7</td>
<td>7</td>
<td>-</td>
<td></td>
<td></td>
</tr>
<tr>
<td>No. of CFD iter.</td>
<td>3191</td>
<td>2802</td>
<td>-</td>
<td>1631</td>
<td>1220</td>
<td>-</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$C_L$</td>
<td>0.6721</td>
<td>0.6687</td>
<td>0.5%</td>
<td>0.5972</td>
<td>0.5936</td>
<td>0.6%</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$C_D$</td>
<td>0.0188</td>
<td>0.0188</td>
<td>0.0%</td>
<td>0.0214</td>
<td>0.0214</td>
<td>0.0%</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$C_m^a$</td>
<td>-0.1115</td>
<td>-0.1122</td>
<td>0.6%</td>
<td>-0.0722</td>
<td>-0.0725</td>
<td>0.4%</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*a pitch moment is related to the quarter point of the wing mean aerodynamic chord

The pressure coefficient contours on the upper wing surface for the Euler flow simulation case at $M = 0.85$ are presented in the Figure 4.2. The figure shows good agreement with the reference simulation.

The plot of maximum span wise wing thickness distribution in Figure 4.3 illustrates that application of essentially two dimensional structural model does not produce any unrealistic geometrical changes of the deformed wing. However, this kind of structural model in combination with RBF transformation method requires use of additional rigid elements in the location of the wing root. If additional elements are not applied, significant change in wing thickness will occur as the result of the non-unique solution of the aero-structural coupling matrix.
Figure 4.2: Comparison of aeroelastic simulation results using wing-box (left half) and beam stick (right half) structural model - surface pressure distribution; Euler flow simulation at $M = 0.85, C_L = 0.5$ (condition 1).

Figure 4.3: Comparison of aeroelastic simulation results using wing-box and beam stick structural models - maximum wing thickness. Euler (left) and RANS (right) simulation at $M = 0.85, C_L = 0.5$ (condition 1) and $M = 0.6, \alpha = 5^\circ$ (condition 2).
4.3.2 Case C: Wing-fuselage geometry, wing-box structural model, $M = 0.85$

The Table 6 presents the computational cost and the resultant forces coefficients. The cost of the aeroelastic simulation is about 4 and 2 times higher compared to the aerodynamic solution in the Euler and RANS cases, respectively.

The drag coefficient in the RANS aeroelastic simulation, in the cruise operating conditions, is about 5.1% higher compared to the aerodynamic solution of the rigid wing. The negative pitching moment, thus the balancing force of the horizontal tail unit, is about 22% higher comparing the same cases.

<table>
<thead>
<tr>
<th></th>
<th>Euler</th>
<th>RANS</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of coupling iterations</td>
<td>Rigid - 20</td>
<td>Elastic - 9</td>
</tr>
<tr>
<td>Total no. of CFD iterations</td>
<td>1790</td>
<td>8683</td>
</tr>
<tr>
<td>$\alpha [^\circ]$</td>
<td>0.72</td>
<td>2.03</td>
</tr>
<tr>
<td>$C_L$</td>
<td>0.5003</td>
<td>0.5009</td>
</tr>
<tr>
<td>$C_D$</td>
<td>0.0117</td>
<td>0.0114</td>
</tr>
<tr>
<td>$C_m$</td>
<td>-0.1683</td>
<td>-0.1628</td>
</tr>
</tbody>
</table>

Table 6: Computational cost and resultant aerodynamic forces coefficients - wing-fuselage geometry, required $C_L = 0.5$

Figure 4.4: Comparison of Euler (upper half) and RANS (bottom half) aeroelastic solution - contours of the surface pressure coefficient distribution at static aeroelastic equilibrium state

Figure 4.4 presents contours of the pressure coefficient on upper surface of the wing-fuselage configuration. The result examination suggests that in both Euler and RANS aeroelastic simulations the wing torsional deformation together with higher required angle of incidence resulted in redistribution of pressure over wing surfaces. The effect of the wing torsional deformation is more obvious in the
sections near the wing tip, where the suction was decreased as a result of the sectional angles of attack decrease (see left part of the Figure 4.5) compared to rigid cases.

![Figure 4.5: Comparison of rigid and aeroelastic solutions - the wing twist angle and the wing loading distribution](image_url)

Plot of the loading distribution over the wing span in the right part of the Figure 4.5 shows that in both cases, Euler and RANS simulations, the wing deformation led to increased loading of the wing inboard part, while the outboard part was alleviated.

### 4.4 Summary

The first focus of the presented test cases was a validation of the computational tool functionality. Different aerodynamic and structural models were used in order to test the tool at various conditions. The considered aerodynamic models were wing-only and wing-fuselage configurations, the fuselage was assumed as rigid. In both cases, the wing-box structural models represented the wing structure. The effect of gravity forces was included considering wing structure mass only. The masses of other components were neglected, although it is obvious that they might influence the static aeroelastic results.

The performed analyses have shown the effect of the wing flexibility on the aerodynamic characteristics and load distribution over the wing. The primary cause of the load redistribution is the wing twist due to aerodynamic forces. The effect of wing weight was an alleviation of the load, as it was expected.

The beam stick structural model was employed in order to evaluate the ability of the computational aeroelasticity tool to handle simplified model of a wing structure. The results show that the simplified model can be employed but adjustment of the model is required. The need for adjustment arises from the properties of fluid-structure interface definition applied in the tool, which is based on the radial basis functions method. There must be some structural nodes which are not in the plane of the model. This requirement was fulfilled by additional nodes, placed in the wing root section, connected to beam node by rigid elements. The cases, employing the beam stick model, also validated proposed method for design of a simplified structure model. The aeroelastic solution using the reference wing-box model was compared with the one employing the beam model. The aeroelastic solutions were compared considering several flow conditions and flow models. The results are comparable in respective cases.

The effect of flow model employed in aerodynamic prediction on aeroelastic solution was evaluated. The aerodynamics was predicted by solving either Euler or RANS equations. In both cases, the aerodynamic forces are calculated from the pressure acting on the wing surface and subsequently interpolated to the structural model. Thus, the main source of dissimilarity between considered cases originates in different pressure distribution, which is caused by the diffusive effects in the viscous flow. In the presented cases, the Euler aeroelastic solution resulted in the higher wing tip deflection than in the RANS solution. Therefore, assuming that the flow predicted by the RANS simulation is closer to the reality, the Euler aeroelastic prediction over-predicts the wing loading in these particular cases.
5 Aerodynamic Shape Optimization of Elastic Wing

5.1 Introduction

An aerodynamic shape optimization using high fidelity flow solvers has been employed for improvement of aircraft aerodynamic design over the last decades. The growing interest in this field was enabled by developments of the Computational Fluid Dynamics (CFD) solvers. CFD solvers became the accepted analysis tools in the aerospace industry reducing the number of tunnel measurements and flight tests during an aircraft development. Due to large number of design variables usually needed for aerodynamic shape design of aircraft, the gradient-based algorithms combined with adjoint solvers are the only meaningful methods for practical application. Locality of those algorithms is a drawback if applied in design space where multiple local optima are likely to occur, the wing shape design is probably such a case. This restricts the process to find only a local optimum near an initial starting point. The solution might be the hybrid optimization algorithms combining non-deterministic (gradient-free) and gradient-based approaches.

The practical application of the gradient-based methods has been probably started by introduction of adjoint sensitivity analysis for Navier-Stokes equation by Pironneau in 1973 [60] and later for incompressible Euler equations [61]. The application in transonic flow regime was enabled by adjoint derivation for compressible Euler equation by Jameson in 1988 [62]. Later he extended the adjoint for Navier-Stokes equations [63]. But the stability and reliability was problematic for long period of time. Nowadays, there are only few adjoint Navier-Stockes solvers applying linearized turbulence models, others rely on approximation by frozen eddy viscosity.

Obviously, the aircraft design is a multi-disciplinary problem. Increased flexibility of aircraft primary structure, as result of modern material application, requirement for lightweight structure and aerodynamically efficient shapes, even emphasizes the multi-disciplinary nature. The wing deformation due to aerodynamic load results, among others effects, in aerodynamic characteristics change. Thus, the performance gain, as result of rigid model optimization, might be decreased or neglected, if applied to real aircraft. The solution might be inclusion of an airframe elasticity to aircraft shape optimization.

In the section, the aerodynamic shape optimization of a common airliner elastic wing is presented and compared with optimization of the same wing assuming rigid structure. The aim is an evaluation of possible benefit and the computational cost of the aerodynamic shape optimization of the elastic wing. A gradient-based optimization approach is applied in connection with adjoint method used for calculation of aerodynamic forces gradients. The computational aerelasticity tool is employed for estimation of the elastic wing aerodynamic characteristics.

5.2 Principle of Aerodynamic Shape Optimization

The scheme of applied aerodynamic shape optimization loop is following. An optimization algorithm directs a decision making in shape design process in order to improve desired aerodynamic characteristics (drag, glide ratio, ...) by minimizing relevant objective function. A parameterization method is employed to describe a given geometry by set of parameters creating a design space. Since the parameterization deforms the surface mesh of the geometry, a mesh deformation tool must be incorporated to propagate the shape deformations into a CFD volume mesh. In the next step, the flow field is solved using either CFD solver, in case of a rigid model optimization, or coupled CFD with Computational Structural Mechanics (CSM) solver, in case of an elastic model. The flow solution provides the values of flow field variables and integral aerodynamic characteristics of the current design.

Gradients of desired variables (drag, lift, moment coefficients), with respect to all surface mesh nodes displacements, are calculated on current shape using adjoint of flow equations solver. In the elastic optimization case, the gradients are calculated on current aerelastic deformed shape. The gradients with respect to design parameters are obtained by multiplication of the surface gradient vector by parameterization Jacobian matrix. The function and gradient values are fed to the optimizer and the loop is repeated until convergence criteria are met.
5.3 Tools

The employed flow and adjoint solver was Edge described in section 2.3.2. The computational aeroelasticity tool (see Section 2.6), coupling the CFD solver Edge with the beam finite element solver introduced in Section 2.4.1, was employed for the static aeroelastic calculation.

A Free Form Deformation (FFD) parameterization based on NURBS was employed for a geometry parameterization. This implementation uses RBF coordinates transformation in order to better control deformations and geometric constraints. Description of the method and a study of numerical properties can be found in [64].

The optimizations were performed by gradient-based optimization algorithm - Sequential Quadratic Programming (SQP) in NLPQLP [65].

The initial CFD grid was created in Ansys ICEM CFD. The spring analogy mesh deformation tool was employed to propagate surface shape changes, resulting from the optimization process, into the CFD volume mesh. For large deformations, such as the wing deformation due to the aerodynamic loading, the RBF mesh deformation combined with the spring analogy method was incorporated.

5.4 Test Cases

5.4.1 Common description

The aerodynamic shape optimization test cases, based on cases proposed in [66], concern drag minimization of the transonic wing of the airliner model (so called Common Research Model [55]) at Mach number $M = 0.85$ and altitude $h = 11000m$. The lift coefficient required for steady horizontal flight in such conditions is $C_L = 0.5$. The formulation of the optimization was:

$$\text{minimize } F(X) = C_D$$
subject to $C_L = 0.5$

\[
C_m \geq -0.1754 \\
V \geq V^{CRM} \\
t_f \geq t_f^{CRM} \\
t_r \geq t_r^{CRM}
\]

fixed trailing edge
wing planform shape fixed

The goal is to decrease the drag while the lift remains constant. The geometrical constraint on the internal volume is meant to ensure minimal space for the fuel. Other constraints on the wing thicknesses at front and rear spar positions, $t_f$ and $t_r$ respectively, are meant to guarantee the same minimal structural height, thus the minimal structural stiffness is ensured. Therefore, the structural stiffness can remain frozen in the elastic wing optimization case.

The constraints on fixed wing planform shape and fixed trailing edge are not explicitly prescribed but they are fulfilled by the choice of optimization variables.

| Table 7: Aerodynamic characteristics of the initial CRM wing |
|-------------|-------------|-------------|
| Lift coefficient, $C_L$ | 0.5000 | 0.5000 |
| Drag coefficient, $C_D$ | 0.0120 | 0.0120 |
| Pitch moment coefficient, $C_m$ | $-0.1750$ | $-0.1717$ |
| Angle of attack, $\alpha$ | 0.721 | 0.816 |
5.4.2 Initial design

Rigid wing optimization The initial geometry for the rigid wing optimization cases was the CRM wing, as it was given in [55]. The CRM geometry was designed to provide common representative model of an airliner operating in transonic conditions for validation of the state-of-the-art CFD solvers. Thus, the geometry of the CRM wing corresponds to the flight shape (1-g shape, bended and twisted due to aerodynamic loading) at nominal cruise conditions at Mach $M = 0.85$ and $C_L = 0.5$ at altitude 12 000 m. Therefore, the subject of the optimization is a rigid wing flight shape at its nominal cruise conditions.

Elastic wing optimization For the elastic wing optimization, it is desirable to use an undeformed wing geometry, as the static aeroelastic analysis determines correct flight shape for given operating condition. Therefore, so called jig shape was designed from the flight shape giving the undeformed wing surface and structural models. The jig shape CFD mesh was created from the mesh of the rigid wing flight shape. Applied structural model is a beam stick finite element model created by 41 beam elements. Each node of the beam element is connected to two additional nodes by rigid elements, which are beneficial for coupling with aerodynamic surface.

5.5 Results

The comparison of the results of the rigid and elastic wing optimization cases is presented in Table 8. The constraint imposed on the value of the lift and the pitch moment coefficients was fulfilled, in both cases. The wing internal volume constraint was satisfied in the rigid wing case. In the other case, the violation was about 0.19%, what might be considered as constraint satisfaction. The optimization objective - the drag coefficient - was reduced by 6.72% and 6.17% in the rigid and elastic wing optimization, respectively. The computational cost in the case of the elastic wing, in terms of number of the flow and the adjoint solutions, is nearly twice as high as in the rigid wing case.

<table>
<thead>
<tr>
<th></th>
<th>$C_{D_{opt}}$</th>
<th>$C_{L_{opt}}$</th>
<th>$C_{M_{opt}}$</th>
<th>$V_{opt}$</th>
<th>Cost$^a$</th>
<th>$C_D$ decrease</th>
</tr>
</thead>
<tbody>
<tr>
<td>Baseline</td>
<td>0.0120</td>
<td>0.5000</td>
<td>−0.1750</td>
<td>84.46</td>
<td>−</td>
<td>−</td>
</tr>
<tr>
<td>Rigid wing</td>
<td>0.0112</td>
<td>0.5000</td>
<td>−0.1754</td>
<td>84.46</td>
<td>84</td>
<td>6.72%</td>
</tr>
<tr>
<td>Elastic wing</td>
<td>0.0113</td>
<td>0.4998</td>
<td>−0.1755</td>
<td>84.28</td>
<td>149</td>
<td>6.1739%</td>
</tr>
</tbody>
</table>

$^a$ Total cost of optimization in terms of number of flow and adjoint of flow solutions

The plots in Figure 5.2 suggest that constraints on the wing thickness were fulfilled in both cases. The thickness at rear spar position remained nearly unchanged in both cases, while the thickness at the front spar was slightly increased near the wing root.
The tuning of the wing shape and the twist angle of the wing sections resulted in redistribution of the lift along wing span. The resultant distribution is closer to ideal elliptical $C_Lc$ distribution, as it is illustrated in Figure 5.3. Moreover, the lift force resultant was shifted towards wing root, what might result in lower bending loading of the wing structure. The same figure shows that the wing twist was changed towards higher negative values in both cases.

The Figure 5.5 presents the pressure coefficient distribution over the upper surface and at chosen wing sections. The plots suggest that in both cases the optimization resulted in nearly shock free solution at the nominal cruise condition. The pressure change is more gradual towards the trailing edge of the optimized wing contrary to the baseline wing with steep increase of the pressure due to shock.

The optimization history plots shown in Figure 5.4 suggest that the objective function decrease was smooth and the optimization criteria were met after 21 iterations in rigid wing case. The peaks

Figure 5.2: Comparison of the wing thickness, at position of the front (left) and rear (right) wing spar

Figure 5.3: Comparison of span-wise wing twist and lift distributions (grey line shows the elliptical distribution of $C_Lc$)

Figure 5.4: History of the optimization
in the plot of the objective function history in case of the elastic wing together with abrupt changes of the optimality imply that there was some source of an error in the optimization chain. The probable source is the neglect of the structural deformation influence on the gradient of the aerodynamic forces and moments. Thus, the implementation of the coupled fluid-structure adjoint equation solver is required.

5.6 Summary

The chapter presents the optimization of the elastic wing compared to the rigid wing optimization. Different initial designs were applied in respective cases for the reasonable comparison. The rigid wing was optimized starting from the flight shape at the design operating conditions. Whereas in case of the elastic wing the initial design was the jig shape, which under loading by aerodynamic forces at the design flight conditions deforms to the same flight shape as in rigid case.

The resultant drag reduction is nearly same in both cases but the computational cost in the elastic wing case is almost doubled. Moreover, the objective function decrease was not as smooth as it might be expected. The several increases of the objective value during the optimization were probably result of inexact gradient calculation. Although the gradients were calculated on the aeroelastic deformed shape, the error in gradient calculation applying pure flow adjoint equation, thus neglecting the influence of the wing structure deformation on the aerodynamic forces gradients, is significant.

Thus, the derivation and implementation of the coupled fluid-structure adjoint equations is a necessity for the further work on the elastic wing optimization. The real benefit of this approach is expected in the multi-point optimization considering more operation conditions.
Figure 5.5: Comparison of surface pressure coefficient distribution - baseline and optimized wings

Baseline flight shape
CD = 0.0120
CL = 0.5000
Cm = -0.1750

Optimized rigid wing
\( \alpha = 0.721 \) deg
CD = 0.0112
CL = 0.5000
Cm = -0.1750

Optimized elastic wing
\( \alpha = 0.816 \) deg
CD = 0.0113
CL = 0.4998
Cm = -0.1755
6 Numerical study of benchmark supercritical wing at flutter condition

6.1 Introduction

Transonic flutter and LCO are two dynamically non-linear phenomena whose prediction is largely dependent on wind tunnel and flight testing. On the computational side, the most used method for aerodynamic predictions in aeroelastic computations is the doublet lattice method [7, 8]. It is essentially a linear method and as such fails predicting non-linear aeroelastic phenomena. Despite the progress in the Computational Fluid Dynamics over the past decades the predictions of the transonic flutter, and LCO and in general, of the non-linear aeroelasticity, remains still a challenge [67, 68, 69]. The computational predictions of the non-linear phenomena are facing several challenges: the fluid-structure coupling, code validation and time synchronization [70]. In his presentation, Bendiksen points out the time synchronization as the most important issue, because a loosely coupled aeroelastic code can give incorrect aeroelastic solution. Similar observations were made in [71, 72, 73, 74] and resulted in the extension of the loosely coupled schemes to a second order accuracy in order to mitigate the time synchronization problem. Apart from the time synchronization problem, other challenges may include the prediction of the flow separation and the flow transition on fluid side and the structural damping on structural side.

This chapter presents the author’s contribution to the research oriented on the assessment of time synchronization for the CFD-CSM coupled problem which was performed by comparing time converged solution of the test case using loosely and strongly coupled fluid-structure interaction. The test case used here is a transonic flow around the Benchmark Super-Critical Wing (BSCW) at flutter condition. Results were compared to the experimental data provided by NASA [75]. A flutter boundary for different Mach numbers was calculated using the proposed method. The results were published online as the journal paper ”Computational Fluid Dynamics Study of Benchmark Supercritical Wing at Flutter Condition” [76].

6.2 CFD-CSM solver

6.2.1 Flow and structural solver

The CFD flow solver used in the study is the Edge solver described in the subsection 2.3.2. The convergence within each time step is controlled by setting a number of minimum and maximum subiterations or by the level of residual reduction. In this study a fixed number of subiterations was specified to get a minimal reduction of the residuals below certain value, usually 2.5 orders of magnitude.

The employed structural solver is part of the Edge solver. It solves differential linear equations valid for a dynamic system with small displacements

\[ M\ddot{x} + C\dot{x} + Kx = f \]  

(6.1)

where \( x \) is the vector of structural coordinates, and \( f(t) \) is the corresponding vector of

6.2.2 Coupling scheme

The coupling scheme is a partitioned coupling scheme. The data between solvers were exchanged on subiteration level. The Figure 6.1 shows a comparison of this scheme with the usual Conventional Staggered Schene (CSS) [71, 72].

6.2.3 Mesh deformation

Since the selected test case has just plunge and pitch modes, the rigid motion (translation and rotation) of the CFD mesh was applied instead of the mesh deformation. The rigid mesh motion is prescribed
using transformation of the modal coordinates to the physical displacement and rotation. This approach replaced the surface deformation, defined as a linear combination of the mode shapes and modal coordinates, and its the propagation to the volume mesh using mesh deformation techniques. The other attribute of the approach is the constant shape of the wing at any displacement.

6.3 Test Case

6.3.1 Experimental Setup

The test case is the Benchmark Super Critical Wing (BSCW), which was experimentally tested at NASA TDT facility [75].

The test case considered in this study is a case of flow around the BSCW wing at Mach number $M = 0.74$, angle of incidence $\alpha = 0^\circ$, Reynolds number $Re = 4.45$ million and dynamic pressure $p = 8082$ Pa ($168$ psf) which is an experimentally measured flutter onset for this wing. The test medium is R-12 coolant gas. The case involves a transonic flow with shock wave on the upper side of the wing. The flow at the angle of attack $\alpha = 0$ degrees is fully attached. The structural model has two modes: the plunging mode with frequency $f = 3.3$ Hz and pitching mode with frequency $f = 5.2$ Hz ([77]). The pivotal point location is at 50% of the airfoil chord - see Figure ??.

6.3.2 Computational setup

The CFD mesh shown consists of 13 millions points and is composed of tetra, prism and penta elements.

The analysis was run in the unsteady Raynold-Averaged Navier-Stokes (URANS) mode using the Spalart Allmaras model [78]. Each solution started with a steady RANS analysis which was subsequently used as an initial guess for URANS analysis of a steady wing. The URANS analysis was run for about 1000 time steps to get the well-converged URANS solution. This solution was then used as an initial guess for URANS coupled aeroelastic analysis. The coupled aeroelastic calculations modeled 5s, in one case 10s, of the physical time.

6.4 Results

6.4.1 Estimate of flutter dynamic pressure

The BSCW wing analysis at the dynamic pressure, which was determined experimentally as a flutter onset dynamic pressure, led to predictions indicating that the wing model is already in flutter. Therefore, the two other lower dynamic pressures were calculated and the final values for the damping ratios and frequency were interpolated and used to estimate the values of the "CFD determined" flutter onset. The Mach number was kept at $M = 0.74$.

Figure 6.2 shows the values of the damping ratio and frequency vs. dynamic pressure.

From the interpolation curves the value of the flutter onset dynamic pressure was estimated to be at $q = 7700$ Pa, which is point where the damping ratio for the pitching motion is zero. The case was
then simulated at this value of the dynamic pressure modeling total of 10s sequence. The final values of the damping ratio and frequency are shown in the Table 9.

<table>
<thead>
<tr>
<th></th>
<th>Pitch</th>
<th>Plunge</th>
</tr>
</thead>
<tbody>
<tr>
<td>Damping</td>
<td>Frequency Hz</td>
<td>Damping</td>
</tr>
<tr>
<td>CFD, $q = 8082Pa$</td>
<td>-0.0034382</td>
<td>4.212</td>
</tr>
<tr>
<td>CFD, $q = 7700Pa$</td>
<td>-0.0000052</td>
<td>4.261</td>
</tr>
<tr>
<td>WT, $q = 8082Pa$</td>
<td>0.0</td>
<td>4.3</td>
</tr>
</tbody>
</table>

Table 9: Results of CFD flutter analysis for pitch and plunge modes compared to the wind tunnel data

Figures 6.3 and 6.4 show magnitude and phase of the pressure frequency response function (FRF) at 60% and 95% of span compared to experimental data.

The comparisons of the magnitude of FRF show fairly good agreement with the slight difference around the leading edge on the lower side of the wing at 60% of span and region of trailing edge at 95% of span. As of now, there is not any explanation what is the cause to the first difference. The second one is most probably caused by the trailing edge separation, which was not accurately modeled by CFD method.

The comparison of the FRF phase shows good agreement of the numerical analysis with the experimental results. The differences observable at the upper sides of the 60% and 95% are just the 360 degree shifts, thus the results are comparable with the experiment. The most significant difference, on the lower side at 95%, might be caused by the problems of the CFD to predict the trailing edge separation.
Figure 6.3: Magnitude in 60% and 95%

Figure 6.4: Phase in 60% and 95%
6.4.2 Estimate of the flutter boundary

The similar procedure as used in section 6.4.1 was used to calculate the flutter boundary for the BSCW wing at angle of attack $\alpha = 0$ degs and range of Mach numbers from $M = 0.6$ to $M = 0.9$. Figure 6.5 shows the comparison of the measured dynamic flutter pressure and flutter frequency vs. Mach number with the values obtained numerically.

![Chart showing comparison of measured and calculated flutter boundary](image)

**Figure 6.5:** Flutter boundary - comparison between CFD and wind tunnel data

The wind tunnel data are available at range of Mach number up to $M = 0.82$ for two gases - air and R-12 gas. The CFD solution is available for R-12 gas only. The figures show strong non-linear dependency of the flutter dynamic pressure on Mach number for Mach numbers larger than $M = 0.82$ which can not be predicted by linear computational methods.

6.5 Summary

This chapter presents numerical study of the Benchmark Super-Critical Wing at a condition where the wind tunnel data indicated a flutter onset. The wing is considered rigid, the aeroelasticity is brought to the system by PAPA apparatus which allows model to rotate around pivotal point and to plunge in the vertical direction. The flow around the wing is transonic without any large areas of the flow separation, which makes the case an ideal test case for validation of the coupled numerical analysis.

The analysis shows that the most important factor influencing the validity of the result is the coupling scheme. The second factor influencing the accuracy of the flutter predictions is the ability of the CFD code to predict various features of the transonic flow including flow separation and the transition of the laminar boundary layer to turbulent boundary layer.

The strong coupled scheme gives result which is changing marginally with different time step. As long as the other factors, which potentially require using very short time steps (such as the flow separation), are not present, the scheme can use of relatively large time step during time integration, as long the sufficient convergence to the pseudo steady state solution within each time step is guaranteed.

The loosely coupled scheme, such as the Serial Staggered Scheme, converged to the different result as the time step of the simulation was refined and there is no indication that with refined time step the results would show asymptotic convergence to the "correct" result.

The subsequent tests with "loosening" the strong coupling scheme by using only several subiteration levels to time synchronization rather then every subiteration showed that such an approach can produce similar results to the strongly coupled scheme as long as there are five or more time synchronization steps during each time step.
7 Conclusions

7.1 Outcome of the Thesis

Computational aeroelasticity tool A tool for simulation and design optimization of the static aeroelastic models has been implemented and tested. It allows to include effect of the static structural deformation of the airframe in the aerodynamic analysis. The implementation is based on the communication between CFD and CSM solvers using I/O operations via hard-copied files. The influence of various settings of the flow solver on the convergence of an aeroelastic solution was tested. The results suggest that optimal settings are case dependent.

Additionally, a numerical structural solver has been designed and implemented and subsequently applied in the computational aeroelasticity tool. It is linear elastic preprocessor and solver for structural models using the beam finite elements. Moreover, it is able to solve static deformation of the finite element model consisting of arbitrary elements. In this case, the stiffness and mass matrices must be provided by an external preprocessor.

Inverse design method for equivalent beam model The method of inverse design of the beam model properties has been proposed and tested. The method finds the beam properties to get equivalent static deformation of the wing to the reference one under the same loading. The results suggest that the proposed inverse method might be applicable for design of simplified structural model of a complex wing geometry.

Numerical study of wing at flutter condition The time synchronization scheme for the coupled CFD-CSM problem was evaluated on the wing flutter test case. The research was conducted within the Aeroelastic Prediction Workshop II in cooperation with colleagues from Swedish Defense Research Agency, FOI. The results were compared with the test case performed at the NASA Transonic Dynamic Tunnel and they are in close agreement with experimental results.

7.2 Conclusion

The objective of the thesis was to design and implement a tool for aeroelastic simulations. The tool should be applicable for the aerodynamic design and analysis of an elastic airplane.

The aeroelastic simulation tool, programmed in the Matlab environment, was designed in the way that it allows to employ arbitrary flow and structural solvers. The fluid-structure transformation interface was defined using the radial basis functions.

The practical applicability of the computational aeroelasticity tool in the aerodynamic analysis was tested on the cases employing different types of the aerodynamic and structural models. The test cases have shown that tool is able to handle complex geometries, such as a wing-fuselage model of transport aircraft with the swept wing.

The application of the designed simulation tool in the aerodynamic shape optimization of the elastic model was evaluated. The optimization problem was defined aiming for a drag reduction of the transonic wing with aerodynamic and geometric constraints. The aerodynamic design optimization was performed for the single operating condition. The results suggest that further development must be performed as the employed gradient calculation neglected influence of the wing deformation due to aerodynamic loading, on the gradients of aerodynamic forces.

7.3 Perspectives

Computational aeroelasticity tool The obvious continuation of the presented work would be the application of the computational aeroelasticity tool for solving time-dependent dynamic aeroelastic problems. In order to do this, the tool must be implemented in the way that the communication between solvers is based on the direct approach via random-access memory. Therefore, the tool as
well as the linear elasticity solver must be programmed in the language such as Fortran or C and implemented into the applied CFD solver Edge.

**Inverse design method for equivalent beam model** There is still space for further development of the method, such as extension for design of beam stick model dynamically equivalent to higher fidelity structural model. It could also solve the suggested problem of indeterminacy of some resultant stiffness characteristics.

**Aero-structural optimization** The result of aerodynamic shape optimization of elastic wing have shown the need for coupled fluid-structure equations adjoint solver. The solver should extend existing flow equation adjoint solver implemented in the CFD package Edge. This extension will allow to perform aero-structural optimization which would increase design efficiency of aircraft from both aerodynamic and structural perspectives.
References


Author’s publications


Author’s Curriculum vitae

PERSONAL AND CONTACT INFORMATION

Name: Ing. Jan Navrátil
Date of birth: 4. 2. 1984
Contact address: Šrámkova 11, 638 00 Brno, Czech Republic
E-mail: navratil@fme.vutbr.cz

EDUCATION

2009 – now  Brno University of Technology
Faculty of Mechanical Engineering, Institute of Aerospace Engineering
Doctoral study programme: Design and Process Engineering
Doctoral Thesis: New Approaches in Numerical Aeroelasticity Applied in
Aerodynamic Optimization of Elastic Wing

2003 – 2008 Brno University of Technology
Faculty of Mechanical Engineering
Master’s study programme: Aeronautical Engineering
Specialization: Aircraft Design
Diploma Thesis: Marabu VUT 001 aircraft conceptual design according to CS-22
regulation

WORK EXPERIENCE

2010 - now  Brno University of Technology
Institute of Aerospace Engineering – research activities, work on project

2008 – 2009 CCI Czech Republic s.r.o.
Design of control valves (steam, water) according EN and ASME regulations

INTERSHIPS

September 2014  Swedish defence research agency – FOI, Stockholm
Research in the field of:

August-October 2013  • Aerodynamic shape optimization,
March –May 2013  • Aeroelastic shape optimization,
April – September 2012  • Fluid-structure interaction
Abstract

The aeroelasticity is an essential discipline involved in the aircraft design, aiming to predict phenomena occurring due to interaction of aerodynamic, elastic and inertial forces. Those phenomena might often lead to catastrophic consequences, thus it must be proven that they do not occur between the speeds bounding the airplane flight envelope.

Current aircraft design leads to increased flexibility of the airframe as a result of modern materials application or aerodynamically efficient slender wings. The airframe flexibility influences the aerodynamic performance and it might significantly impact the aeroelastic effects, which can be more easily excited by rigid body motions than in case of stiffer structures. The potential aeroelastic phenomena can occur in large range of speeds involving transonic regime, where the non-linear flow effects significantly influence the flutter speed. Common aeroelastic analysis tools are mostly based on the linear theories for aerodynamic predictions, thus they fails to predict mentioned non-linear effect.

The objective of the thesis is, therefore, to design, implement and test an aeroelastic computational tool employing the aerodynamic prediction solver which is able to predict non-linear flow. In the thesis, the main focus is directed to the static aeroelastic simulations.

The methods involved in numerical static aeroelastic simulation are presented in the thesis. The implementation of the computational aeroelastic tool was described and the convergence of the coupled solver was investigated. The tool functionality was validated in the various test cases. The tool was applied also in the aerodynamic shape optimization of an elastic wing. The results and computational cost were compared to the rigid wing optimization.

Last chapter presents the author’s contribution to the research oriented on the assessment of time synchronization scheme for the CFD-CSM coupled problem. The test case used here is a transonic flow around the Benchmark Super-Critical Wing at flutter condition. Results were compared to the experimental data provided by NASA.

Abstrakt

Aeroelasticita je nezbytná vědní disciplína zahrnutá do návrhu letounů. Zaměřuje se na předpovídání jevů, které vznikají vlivem interakce aerodynamických, elastických a setrvačných sil. Tyto jevy často vedou ke katastrofickým následkům, proto musí být prokázáno, že nevzniknou v rozsahu rychlostí ohraničujících letovou obálku.

Aplikace moderních materiálů při konstrukci draku, spolu se snahou navrhnout aerodynamicky efektivní tvar krídla, vede ke zvyšování poddajnosti letounů. To má za následek změnu aerodynamických vlastností a také k výraznějšímu vlivu na aeroelasticke jevy, které mohou být vyvolány snadněji vlivem pohybu tuhého tělesa než v případě tužších konstrukcí. Aeroelasticke jevy mohou vznikat v širokém rozsahu rychlostí zahrnujícím i transsonickou oblast. V této oblasti je ovlivněna zejména rychlost, při níž dochází k třepění, a to vlivem nelineárních jevů v proudu. Běžné nástroje, které jsou založeny na lineárních teoriích, nejsou schopny tyto nelineární jevy popsat.

Cílem práce je proto navrhnut a implementovat a otestovat nástroj pro výpočet (numerickou) simulaci aeroelasticity. Nástroj má využívat řešit proudového pole, který je schopen předpovědět nelineární jevy. V práci je kladen důraz na simulaci statické aeroelasticity.

V práci jsou popsány metody, které je nutno zahrnout do numerické simulace statické aeroelasticity. Dále je popsán vlastní nástroj a je provedeno zhodnocení konvergence statických aeroelastických výpočtů. Funkčnost nástroje byla ověřena na příkladech, kdy byly použity různé aerodynamické a strukturální modely. Nástroj byl také aplikován při aerodynamicke tvarové optimalizaci poddajného krídla. Výsledky optimalizace a její výpočetní náročnost byly porovnány s případem optimalizace tuhého krídla.

Na závěr je v práci prezentován příspěvek autora do výzkumu zaměřeného na zhodnocení vlivu časové synchronizace mezi CFD a CSM řešiči. Použitý testovací případ je transsonické obtékání krídla na začátku třepetání (flutteru). Výsledky byly srovnány s experimentálními daty poskytnutými NASA.