Elements of Automated
Aeroelastic Analysis
in Aircraft Preliminary Design
Elements of Automated Aeroelastic Analysis in Aircraft Preliminary Design

Proefschrift

ter verkrijging van de graad van doctor aan de Technische Universiteit Delft, op gezag van de Rector Magnificus prof. dr. ir. J.T. Fokkema, voorzitter van het College voor Promoties, in het openbaar te verdedigen op maandag 11 juni 2007 om 15.00 uur

door

Paolo LISANDRIN

Ingegnere aeronautico, Università degli Studi di Roma “La Sapienza”, Italië, geboren te Latina, Italië
Dit proefschrift is goedgekeurd door de promotoren:

Prof. dr. ir. M.J.L. van Tooren
Prof. ir. A. Beukers

Samenstelling promotiecommissie:

Rector Magnificus, Voorzitter
Prof. dr. ir. M.J.L. van Tooren, Technische Universiteit Delft, promotor
Prof. ir. A. Beukers, Technische Universiteit Delft, promotor
Prof. dr. ir. Z. Gurdal, Technische Universiteit Delft
Prof. dr. ir. D. Rixen, Technische Universiteit Delft
Prof. dr. M. Gennaretti, Università RomaTre, Roma, Italy
Dr. W.A. Silva, NASA Langley Research Center, Hampton, VA, USA


Keywords: Aeroelasticity, Finite Elements, P-Elements, Modal Analysis, Computational Fluid Dynamics, Transonic Aerodynamics, Transonic Flutter, System Identification, Parametric Modelling, Reduced Order Models, Composite Materials, Stiffened Panels, Knowledge Based Engineering

Copyright © 2007 by Paolo Lisandrin

All rights reserved. No part of this publication may be reproduced, stored in a retrieval system or transmitted in any form or by any means, electronic, mechanical, photocopying, recording or otherwise, without the prior written permission of the author P. Lisandrin, Delft University of Technology, Faculty of Aerospace Engineering, P.O. Box 5058, 2600 GB Delft, the Netherlands.

Printed in the Netherlands
## Contents

1 Introduction

1.1 The problem .......................................................... 1
1.2 Goal of the thesis ..................................................... 3
1.3 The equation of the aeroelasticity .................................. 5

2 P-formulation Finite Elements for Structural Dynamics .... 11

2.1 Introduction ............................................................. 11
2.2 Fundamentals of the p-formulation finite element method . 14
2.3 Hierarchical shape functions of quadrilateral elements .... 17
2.4 Hierarchical shape functions of hexahedral elements .... 18
2.5 Nastran implementation of p-elements ......................... 20
2.6 Modal analysis of flat plates ....................................... 20
2.6.1 Thick plate ............................................................ 21
2.6.2 Thin plate ............................................................. 26
2.7 Modal analysis of curved panels .................................. 26
2.7.1 Thin cylindrical panel .............................................. 28
2.7.2 Thin spherical panel ................................................. 30
2.7.3 Mass matrix formulation .......................................... 32
2.8 Discussion ............................................................... 33

3 Solid P-Formulation Finite Elements for Composite Structures 35

3.1 Introduction ............................................................. 35
3.2 Solid orthotropic model for composite plates ................. 35
3.3 Applications ............................................................ 37
3.3.1 Unconstrained Facing Analysis ................................. 38
3.3.2 Unconstrained flat sandwich panel ............................ 40
3.4 Conclusions and recommendations ............................... 45

4 Equivalent Models for Stiffened Panels ......................... 47

4.1 Introduction ............................................................. 47
4.2 Global stiffness determination for stiffened panels ......... 48
4.2.1 Orthotropic materials .............................................. 49
4.3 Equations for the equivalent modelling ......................... 53
4.3.1 One layer smeared stiffener approach ....................... 54
4.3.2 One layer smeared stringer simplified approach .......... 56
# Contents

4.3.3 Mass Equivalence ........................................... 58
4.3.4 Shear stiffness equivalence ................................ 58
4.4 Wing-box analysis .............................................. 62
4.5 Conclusions and recommendations .......................... 66

5 Modal Analysis of Wing-Like Structures ..................... 69
  5.1 Introduction .................................................. 69
  5.2 Solid geometry generator ................................... 70
  5.3 Solid mesh generator ......................................... 74
  5.4 Applications .................................................. 80
     5.4.1 Straight wing ........................................... 80
     5.4.2 Blended wing body ...................................... 85
  5.5 Conclusions and recommendations ........................ 101

6 System Identification for Flutter Analysis .................... 103
  6.1 Introduction .................................................. 103
  6.2 NACA64A010 test case ....................................... 105
  6.3 MBBA3 test case .............................................. 121
  6.4 RAE2822 test case ........................................... 121
  6.5 Flutter results ............................................... 122
  6.6 Conclusions .................................................. 127

7 Conclusions ....................................................... 129

8 Recommendations ................................................ 131

A System Identification Tool for a DEE ......................... 133
  A.1 Definition of the input function ......................... 133
  A.2 Parameters needed for the system identification procedure 136
  A.3 Mesh generation for a system identification tool .... 136

Bibliography .......................................................... 141

Summary ............................................................. 149

Samenvatting ......................................................... 153

Publications ......................................................... 157

Acknowledgements .................................................. 159

About the Author ................................................... 161
Chapter 1

Introduction

1.1 The problem

In most cases the overall aircraft design process can be subdivided into three parts: conceptual, preliminary and detailed. Different from the detailed design phase, where a configuration has already been defined and prototyped, the most crucial phases for the successfulness of the project are the conceptual and the preliminary design phases. In the conceptual design phase several eligible configurations are generated according to the requirements of the design and the design process is diverging, Fig. 1.1. In the preliminary design phase, each configuration selected during the conceptual design phase is analysed with respect to different disciplines such as aerodynamics, structures, stability and control, costs, manufacturing. The more disciplines involved in the preliminary design phase, the better the chance to select the best configuration fulfilling all the requirements.

At this stage none or limited experimental data are available to the designer, and therefore limited verification or update of the numerical models is possible. Errors made during this phase and recognized at a later stage, can lead either to an expensive redesign of the complete concept or to the introduction of major modifications (weight penalties, flight envelope limitations) of the final configuration. Therefore, even if several configurations are selected during the conceptual design phase, the need of reliability in the results often brings to discard novel designs. Lacking experimental data, the use of numerical tools based on sophisticated mathematical models (for example Navier Stokes equations for the aerodynamics) verified with available test cases could improve the reliability of the numerical results, but on the other hand it would slow down the analysis process.

In addition, the multidisciplinary nature of the preliminary design phase requires the introduction of frequent changes in terms of geometrical and physical properties of the design (structural lay-out, shapes, and materials for example) in order to fulfill all the requirements for the different disciplines. This translates in doing many repetitive calculations which, because of the time constraints, often leads to the use of low fidelity and sometimes oversimplified models, in contrast
to the need of high fidelity tools to balance the lack of experimental data. This defines a first issue at this stage.

Another issue is represented by the fact that all the different models used for the various disciplines are usually derived from a unique CAD model from which “customized” models are built for each discipline. The fragmentation makes the update of these models cumbersome and time consuming as the changes are generally implemented by hand, reducing the number of iterations of the preliminary design phase to a minimum. This makes very cumbersome the implementation of an effective multidisciplinary design environment.

In order to overcome the issues raised above and in order to make the preliminary design cycle more effective, during the last years efforts have been spent in the automation of the operations performed during the preliminary design phase. The attention has been focused initially on the development of parametric modeling capabilities of the CAD environment in order to automate all the operations regarding the changes in the geometric properties of the model.

At this level the bottleneck is given by the possibility to translate the changes in the geometric properties into changes in the models generated for the different disciplines, and update these models during the several iterations required during the preliminary design phase.

A possibility to overcome the last issue is offered by Knowledge Based Engineering (KBE) techniques. KBE is an engineering method in which knowledge about the product, e.g. the techniques to design, analyze and manufacture a product, are stored in the product model. The product model represents the engineering intent behind the geometric design. It captures the How and Why, in addition to the What of the design. It can store product information - attributes of the physical product such as geometry, material type, functional constraints - as well as process information - the processes by which the product is analysed, manufactured and tested. It captures the design strategy required to produce a particular product from a specification. It is the set of engineering rules (not only rules involving the geometry) used to design the product.

The KBE approach has been used to build part of a Design and Engineering Engine (DEE), currently under development in the DAR group of Delft University of technology. The DEE consists of a set of properly interconnected toolboxes such that automated multidisciplinary design, analysis and optimisation becomes feasible. The toolboxes are of different nature and include different pieces of analysis software, interface blocks etc. Core element of the DEE is a (multi-) model generator in which the parametrical description of the product resides. It gets input (a parameter set) from a concept/variant generator and generates/regenerates different models (aerodynamic, structural, costs etc) for the analysis tools (the discipline silos). The models consist of geometrical surfaces, numerical values, ASCII files etc. to feed the different analysis boxes (FEM packages, CFD codes etc). The output generated by the analysis tools is given back to an evaluator/optimiser that on the basis of some target function gives a response and issues a new set of input data for the model generator. This new input for the model generator can be fed through the concept generator, whenever a different subsystem concept (aerodynamic, structural) based on
updated parameter values must be instantiated or whenever even a completely
different aircraft configuration is needed for the next iteration step. The general
architecture of the DEE and the design of the single toolboxes need to be flexible
to handle a wide range of different design problems with a minimum of modi-
fication required. No built-in design decisions in general should be hard-coded
inside the toolboxes. The DEE is not intended to replace the functionality of
property software tools, but to integrate them in a wider organized system.

1.2 Goal of the thesis

The concept of the DEE can be used to support aircraft’s aeroelastic analysis.
The discipline silos of such a DEE can be thought as made of two separate blocks,
Fig. 1.3: a structural block taking in input (aerodynamic) forces and giving as
output displacements, and an aerodynamic block taking as input displacement
and velocities (for example to set the impermeability boundary conditions in
the inviscid case) and returning as output aerodynamic forces. Typically the
structural block is a finite element code, whereas the aerodynamics block consists
of an aerodynamic solver.

The current approach followed to perform an aeroelastic stability analysis
(flutter, divergence) consists first in deriving a finite element model for the struc-
ture (assumed linear throughout this work) and second determining the vibra-
tion modes. The modes are used as a base for the reduction of the finite element
model, expanding the displacement field as a function of the modal shapes. The
modal representation adopted for the structure, plugged in the structural block,
results in the replacement of the aerodynamics block with the Generalized Aero-
dynamic Forces (GAF) matrix which is the transfer function between the modal
displacements and the generalized aerodynamic forces (projection of the aero-
dynamic pressures on the modal base chosen). The modal approach makes the
aeroelastic analysis very efficient from the computational point of view and is
therefore particularly suitable for preliminary design analysis.

In the common industry practice the structural models used for aeroelas-
tic analysis often consist of “stick” models, based on beam finite elements and
lumped masses conveniently placed, derived from a very detailed FE model.
Each time a modification is introduced in the properties of the configuration
analysed, new reduced models must be generated. The reduction process is nei-
ther straightforward nor simple. Using a full finite element model would save all
the additional operations connected with the derivation of a reduced model each
time a new configuration is generated, together with the interfacing operations
with the geometrical model, and would keep a tighter and more consistent cor-
relation between the finite element model and the original geometrical (CAD)
model. On the other hand using a full finite element model can cause difficul-
ties in the convergence checks. Indeed in the common practice convergence is
achieved increasing the number of elements in the model. An automated mesh re-
finement process would be best suitable for the convergence checks for the analy-

\footnote{In the hypothesis of small perturbations for the flow equations.}
sis of a full aircraft. Defining a criterion for the automatic mesh (re-)generation based on the evaluation of the FE solution is not an easy task, and this is quite important when few or none experimental data are available and the computations rely mainly on numerical models.

The aerodynamic flow model used for aeroelastic stability analysis in transonic regime, must take into account the presence of shock waves. As a result, additional non-linear terms appear in the flow equations. Traditional aerodynamic analysis methods, like the Doublet Lattice for example, are no longer applicable. In this case Full Potential or Transonic Small Disturbance equations can be used, but these models are based on very restrictive assumptions on the shock strength (isentropic flow). Therefore their range of applicability is limited to conditions for which the entropy loss are negligible or none, i.e. weak shocks. On the other hand, the use of a Computational Fluid Dynamics (CFD) code can give the freedom to analyse any kind of shock strength and shock motion around a steady flow condition. Nowadays the appearance of parallel solvers, the use of unstructured solvers – which allows a high level of flexibility in modeling complex shapes – make CFD tools attractive for stability analyses. Nevertheless a direct coupling of a CFD code with a structural analysis code, in a time-marching sense, is certainly not suited in a preliminary design environment. By making use of system identification techniques, the CFD code can be replaced by a Reduced Order Model (ROM), which is a system having less (finite) number of degrees of freedom. The ROM is described in terms of parameters depending on the transonic steady-state configuration (identified by the Mach number and angle of attack) about which the stability analysis is performed. By using a ROM for the aerodynamics block, the aeroelastic analysis becomes computationally efficient. ROMs are intended to represent the dynamics of the (aerodynamic) system around a reference configuration and they can be derived using either linear or non-linear system identification techniques. The possibility to use a linear system identification technique is appealing because building a ROM requires a number of CFD simulations which are reduced to a minimum whenever a linear ROM can be employed.

The goal of this thesis is twofold:

1. Introduce a finite element approach, suitable to be embedded in a KBE environment, that makes use of a full finite element model to derive the modes of a structure, eliminating all the operations connected with the use of “stick” models and overcoming the inconvenient coming from the use of finite elements based on linear, fixed order, shape functions (Chapters 1-5);

2. Investigate the conditions for which a ROM suitable for aeroelastic analysis in transonic range and in a preliminary design context can be derived using a linear system identification procedure (Chapter 6, Appendix 1).

Non-linear terms are present also in the subsonic and supersonic flow equations. For example, considering a potential flow, non-linear terms appear in the Bernoulli equation (square of the velocity) and in the impermeability condition (the normal to a surface moving in the flow).
1.3 The equation of the aeroelasticity

The equation of an aeroelastic system can be recast as [17]:

\[ \rho \frac{d^2 v(x, t)}{dt^2} + L v(x, t) = f(x, t), \]  

(1.1)

where \( v \) is the displacement field, \( L \) is the structural operator, \( \rho \) the density, \( f \) the (body) forces, \( x \) is the position vector, \( t \) is the time. The problem formulation is completed by adding the initial and boundary conditions. The aerodynamic (surface) forces enter in the Eq. 1.1 through the boundary conditions. Nevertheless they can be introduced in the volume forces vector \( f \) by making use of a Dirac function and the boundary condition for the problem defined by Eq. 1.1 become homogeneous. Throughout the present work, it is assumed that the density is constant in time and space and the structural operator is linear. In addition, no inputs are considered, i.e. no gravity forces are applied. The equation can be solved numerically applying Galerkin’s method, which consists in seeking an approximate solution of the type:

\[ v = \sum_{n=1}^{N} u_n(t) \Psi_n(x) \]  

(1.2)

(where \( \{ \Psi_n \} \) is a set of linearly independent functions satisfying the boundary conditions, \( N \) being the total number of functions chosen), and by substituting the proposed expression in the Eq. 1.1. Then denoting

\[ < a, b > := \int_{V_S} a(x)b(x)dV \]  

(1.3)

the projection of the vectorial function \( a \) on the vectorial function \( b \) in the (volume) domain \( V_S \), the Galerkin method consists in the projection of Eq. 1.1 on any of the independent functions \( \Psi_k \), \( (k = 1 \ldots N) \):

\[ < \rho \ddot{v}, \Psi_k > + < L v, \Psi_k > = < f, \Psi_k > \]  

(1.4)

and then, substituting the expansion 1.2 in Eq. 1.4 yields:

\[ < \rho \dot{u}_n, \Psi_k > + < L u_n, \Psi_k > u_n = < f, \Psi_k >, \]  

(1.5)

where the standard summation convention has been adopted and therefore the repetition of the index \( n \) in each term of the equation above implies a summation with respect to the same index. In compact form:

\[ M_{kn} \ddot{u}_n + K_{kn} u_n = f_k \]  

(1.6)

Throughout this work the only surface forces are the pressures because the fluid model taken into account will be an inviscid model.

For a thorough discussion about satisfying the natural and essential boundary conditions see also Strang and Fix [18] or Reddy [19].
where
\[ M_{kn} = \int_{V_S} \rho \Psi_k \cdot \Psi_n dV = M_{nk} \text{ (Mass matrix)}, \]
(1.7)
\[ K_{kn} = \int_{V_S} \Psi_k \cdot L \Psi_n dV = K_{nk} \text{ (Stiffness matrix)}, \]
(1.8)
\[ f_k = \int_{S} f \cdot \Psi_k dS \text{ (Generalized Forces vector)}. \]
(1.9)

The symmetry of the stiffness matrix comes from the fact that the structural operator is self-adjoint \[17\].

Equation (1.6) defines a system of ordinary differential equations, which can be recast in a different form using the modal transformation:
\[ u = Z q, \]
(1.10)

\( Z \) being the matrix of the eigenvectors obtained from the solution of the structural problem:
\[ \mathbf{M} \ddot{u} + \mathbf{K} u = \mathbf{0}. \]
(1.11)

Left-multiplying Eq. (1.6) by the transpose of the eigenvector matrix \( Z^T \) and using the transformation (1.10), Eq. (1.6) becomes:
\[ Z^T M \ddot{q} + Z^T K Z q = Z^T f. \]
(1.12)

Exploiting the eigenvector orthogonality properties with respect to the mass and stiffness matrices, and normalizing the eigenvectors with respect to the mass matrix we get:
\[ I \ddot{q} + \Omega^2 q = q_D e, \]
(1.13)
\( I \) being the identity matrix, \( \Omega^2 \) the diagonal matrix of the natural (circular) frequencies of vibration of the structure, \( q_D = \rho_\infty U_\infty^2 / 2 \) the dynamic pressure, \( e \) is the projection of the generalized forces vector on the eigenvectors considered. It can be shown \[17\] that this operation is equivalent to projecting the aerodynamic pressure on the approximate eigenfunctions of the operator \( L \). The eigenfunctions are given by the combination of the eigenvectors with the linearly independent functions used in the approximation of the displacement field. The advantage in solving Eq. (1.13) instead of Eq. (1.6) is that the size of the system (1.13) is much smaller than the system of equations (1.6). Although the number of eigenvectors obtained solving Eq. (1.11) equals \( N \), only \( M << N \) are needed in the transformation (1.10) to obtain a converged solution with a limited loss in accuracy; typically, \( M = O(10) \) whereas \( N = O(10^4 \div 10^5) \). The reason is that the transformation (1.10) is equivalent to expanding the displacement field \( v \) in terms of the natural modes of vibration of the structure \[17\]. The modes are orthogonal functions which guarantee a convergence rate higher than the convergence rate obtained expanding the displacement field in terms of a general set of independent functions.

\[^5\] \( \rho_\infty \) and \( U_\infty \) are respectively the fluid density and velocity in the undisturbed flow condition. The expression given for \( q_D \) is valid for airplanes (fixed wings) only.
The equation of the aeroelasticity

For linear flow models, the Laplace transform of the generalized force vector can be expressed as:

\[ \tilde{e} = \mathbf{E}(s l / U_\infty) \tilde{q}, \]

where \( \mathbf{E} \) is the Generalized Aerodynamic Force (GAF) matrix which is a transcendental function of the Laplace variable \( s \), through the complex reduced frequency \( p = s l / U_\infty \), \( l \) being a reference length (for example the chord for a wing section) and \( U_\infty \) the freestream velocity. An equation of the same type of Eq. 1.14 can be obtained for rotors in hover \[21\] and propellers \[22\]. The matrix \( \mathbf{E} \) can be described analytically for simple aerodynamic theories otherwise it can be evaluated numerically, for instance by the doublet lattice method \[17\] or the boundary element method \[17\]. In the Laplace domain the aeroelastic equation appears as:

\[ [s^2 \mathbf{I} + \Omega^2 - q_D \mathbf{E}(s l / U_\infty)] \tilde{q} = \mathbf{0}. \]

This equation defines a non-linear eigenvalue problem in the variable \( s \). In the simplest case it can be solved using an iterative procedure like the p-k or V-g method for example, providing either the flutter or the divergence velocity for an aeroelastic system.

It must be remarked that the numerical form of the aeroelasticity equation is based on two major hypotheses which, as already stated above, are also the focus of the present thesis:

1. Linearity of the structural operator which allows the modal transformation
2. Linear(-ized) flow equations which allows the definition of the GAF matrix, Eq. 1.14
Figure 1.1: The typical diverging/converging design process.

Figure 1.2: Structure of a Design and Engineering Engine.
1.3. The equation of the aeroelasticity

Figure 1.3: Staggered view of an aeroelastic analysis tool.
Chapter 2

P-formulation Finite Elements for Structural Dynamics

2.1 Introduction

The Finite Element (FE) method is one of the standard analysis tools used in the industry. One fundamental step in the aeroelastic analysis of an aircraft is the calculation of the modal shapes and the natural frequencies.

In order to exploit the flexibility offered by a KBE approach to multi-model generation, a FE modelling and analysis tool should support:

- complete automatic FE geometric pre-processing, limiting to a minimum the interaction with the user;
- fast and accurate calculations using of a full model, to overcome the approximations introduced by the use of beam reduced models.

To understand how to achieve these goals, we should consider the two main problems faced by an automatic mesh generator when a FE model is created: the consistency and the convergence of the model.

1. Consistency is related to the type of finite element to be used for modeling a certain structural element. The library of most finite element codes offer a wide range of finite elements, each of them with its own field of application and its limits. Each of them is guaranteed to be applicable for certain applications and is suitable for modeling specific structural elements. An experienced FEM analyst knows which kind of element to place where and why. Obviously an automatic mesher should be able to do the same or circumvent the problem. A possible approach to have an automatic mesher matching the right solution would be the use of a “general” element, capable of representing the behavior of 1-D, 2-D as well as 3-D structural elements.

2. Convergence of the model, is related to the choice of the number of elements necessary to achieve the required degree of accuracy of the solution.
In the current practice the acceptance of convergence is left to experienced people, but in an automatic mesh generator/analysis tool where every decision should be ruled based, a criterium is needed in order to define this number. In theory a convergence analysis should be performed, for example increasing the number of elements within the structural component analysed. Commercially available mesh generators offer the possibility to automatically mesh components based on a minimum element size. This approach could be also applied to a complete aircraft, but the regeneration of the mesh could be quite cumbersome, and the mesh (re-)generation should adapt to the regions of higher gradients. The solution could be the use of elements having an easier approach in the convergence verification.

A solution to both problems of consistency and convergence can be found in the use of finite elements based on higher order shape functions, in the form of the so-called “p-formulation”, “brick” element type.

P-formulation finite elements have been introduced by Babuska and Szabo [23]. For this class of finite-elements convergence is achieved increasing the polynomial order of the shape functions, built through the use of so-called hierarchical polynomials [23], such that everytime the order of a polynomial is increased new degrees of freedom are added.

In the common industrial practice different types of finite elements are used to model a structure. Each type has its own capability and validity range. From the point of view of automation of the calculation and the modeling errors an advantage could be the modeling of the complete structure using only one type of finite element, the aforementioned brick, p-formulation.

A first advantage of the proposed approach stems from the use of a finite-element based on a fully 3-dimensional formulation for the displacement field. The lack of any a-priori assumptions on the kinematics or the geometry of the structure, in contrast with formulations based on thin plate theory [24], makes this element suitable for representing any kind of structural part i.e. both thin and thick shells and beams [25]. Therefore, only one kind of finite-element can be used to model and analyse a complete structure, with the potential of bringing a major improvement in speed and flexibility in the first modeling phase of the structural analysis or when modifications in the structural model are introduced and a new finite element model must be generated. This improvement is particularly important in the preliminary design where a big amount of time is spent in the preparation, modification, and updating of the models to follow modifications in the design. These operations, very time consuming and normally performed by hand, restrict the time dedicated to the design phase.

By using only one kind of finite element the meshing phase can be easily automated. No choice between different elements is needed thus reducing to a minimum the modeling errors.

Solid p-version finite elements have been used for modelling simple thin walled structural elements and solving static problems [25], [26]. Other authors introduced the solid-shell elements, a class of finite element models intermediate between thin shell and conventional solid elements for modelling a wide range
2.1. Introduction

of thicknesses (from thin to moderately thick). Solid-shells have the same node and degrees of freedom configuration of solid elements but account for shell-like behavior in the thickness direction. Still the shell-like behavior limit the applicability of these methods as the occurring of the locking phenomenon must be prevented. For example a “solid plate” finite element for modeling static and fracture behavior of thin shells structures made of composite materials has been developed by Hashagen and de Borst [27] with the behavior along the thickness of the plate limited to a quadratic approximation. Other examples can be found in Felippa [28] or in many commercial packages.

Another advantage of the present approach comes from the use of a finite element based on high-order shape functions, in the form of the p-formulation. It is known that with the commonly used linear interpolation shape functions, locking phenomenon [29], [30], [26] can occur. It is due to the inability of the lower order shape functions to correctly interpolate the displacement field, leading to an overestimation of the stiffness of the element. Several remedies, such as selective and reduced integration, incompatible modes, assumed strains [31], are implemented in commercial finite element codes to avoid locking. In the present work the use of high order shape functions directly eliminates the occurrence of this phenomenon without recurring to any numerical treatment.

A third advantage stems from the fact that in the p-elements the order of the shape function is not fixed; it can be varied in principle to any value. Adaptive analyses can be performed, deciding a starting order for the polynomials to be used and increasing it until a certain convergence criterion has been satisfied or when a maximum polynomial order has been reached. Convergence checks on complex structures such as a whole aircraft becomes feasible whereas by using common linear elements, or using other higher order formulations where the order of the polynomial is fixed, as in Morino et al. [32], this wouldn’t be possible without regenerating the complete mesh several times. In the industry practice often just one and final mesh is generated with the number of elements defined a-priori by experienced people, or, especially for modal analyses, reduced models made of beams and lumped masses are used. These models are not always affordable, especially in structures made of composite materials where the reduction process is even more difficult than structures made of isotropic materials. P-formulation elements make the convergence check process easily automatizable and this is particularly important in the Preliminary Design/MDO environment where no experimental data are available and product properties derivation relies heavily on the numerical calculations.

In the aerospace engineering field, p-formulation finite elements have not received much attention. P-elements are available in some commercial packages, for example in the finite element package Nastran and in the finite element module of the CAD package Pro/Engineer. In these packages the use of the p-elements is shown for stress analysis of details, for the dynamic analysis of topologically three-dimensional structural elements (turbine blades). No application has been done so-far to support a KBE approach to the design of an aerospace structure, which is one of the objectives of this thesis.
2.2 Fundamentals of the p-formulation finite element method

The determination of the modes and frequencies for the structural system represented by the equation (introduced in section 1.3):

\[ \rho \frac{d^2 \mathbf{v}(\mathbf{x}, t)}{dt^2} + \mathbf{L} \mathbf{v}(\mathbf{x}, t) = 0 \]  

(2.1)

is done applying the Galerkin method (see section 1.3), where the \( \Psi_n(\mathbf{x}) \) are the shape functions of the finite element method.

In the classical approach of the finite element method the \( \Psi_n(\mathbf{x}) \) are local interpolation functions of the values of the unknown \( \mathbf{u} \) at the grid nodes. In one dimension, the standard finite element shape functions are defined by the set of the Lagrange polynomials of order \( p \), defined over the domain \(-1 \leq \xi \leq +1:\)

\[ \Psi_i(\xi) = \frac{p+1}{\prod_{j=1, j \neq i}^{p} (\xi_i - \xi_j)} \xi - \xi_i \xi_i - \xi_j, \quad i = 1, 2, \ldots, p + 1. \]  

(2.2)

For each polynomial degree \( p \) chosen for the shape functions a new set of functions must be generated, see Fig. 2.1a, and new stiffness and mass matrices must be determined.

In the \( p \)-version of the finite element method \[26\] the functions \( \Psi_n(\mathbf{x}) \) are the hierarchic shape functions built through the set of Legendre (orthogonal) polynomials. The main difference between the standard and the hierarchic shape functions is that in the hierarchic case all lower order shape functions are contained in the higher order basis. The set of one-dimensional hierarchical shape functions, introduced by Szabo and Babuska \[33\] is given by (see Fig. 2.1b for an example):

\[ \Psi_1(\xi) = \frac{1}{2}(1 - \xi), \]  

(2.3)

\[ \Psi_2(\xi) = \frac{1}{2}(1 + \xi), \]  

(2.4)

\[ \Psi_i(\xi) = \phi_{i-1}(\xi), \quad i = 3, 4, \ldots, p + 1, \]  

(2.5)

where \( \phi_i \) is a polynomial function of maximum degree \( i \), with

\[ \phi_i = \frac{1}{\sqrt{4i^2 - 2}} (L_i(\xi) - L_{i-2}(\xi)), \quad i = 2, 3, \ldots, p, \]  

(2.6)

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{fig2.png}
\caption{Set of one dimensional Standard and Hierarchic shape functions for \( p = 1, 2, 3 \) (from \[26\]).}
\end{figure}
2.2. Fundamentals of the p-formulation finite element method

$L_i(\xi)$ being the Legendre polynomials. The linear functions $\Psi_1(\xi), \Psi_2(\xi)$ are called \textit{nodal shape functions} as they coincide with the linear shape functions based on the Lagrange polynomials. Since

$$\Psi_i(-1) = \Psi_i(1) = 0, \quad i = 3, 4, \ldots, p + 1, \quad (2.7)$$

the functions $\Psi_i(\xi) (i = 3, 4, \ldots, p + 1)$ are called \textit{internal shape functions} and the degrees of freedom associated are the so-called \textit{internal} degrees of freedom. The use of the hierarchic shape functions has an immediate consequence on the structure of the mass and stiffness matrices. If the equations are ordered in such a way that all linear shape functions are numbered from 1 to $n_1$, all the quadratic shape functions are numbered from $n_1 + 1$ to $n_2$ and so on, the stiffness and mass matrices corresponding to a polynomial order from 1 to $p - 1$ are sub-matrices of the stiffness matrix corresponding to polynomial order $p$. Figure 2.2 shows the structure of a matrix and a load vector corresponding to polynomial degree of $p = 3$. The advantage of the p-version finite element method over a higher order finite element method based on standard shape functions is clear, as increasing the order of the shape functions, the hierarchical shape functions allow the reusability of all existing elements of the stiffness and mass matrices in the next solution. Moreover, it has been shown also that increasing the polynomial order of the shape functions, the condition number of stiffness and mass matrices remains constant and is much lower if hierarchical rather than standard shape functions are used, both for two and three dimensional static problems [34]. Adaptive analyses becomes feasible at an affordable computational costs even for complex configurations. As an example the hierarchic shape functions used for an approximation of the displacement field up to the

![Hierarchic structure of the stiffness matrix and load vector for p = 3](from [26]).
third polynomial order are elaborated below:

\[
\begin{align*}
\psi_1(\xi) &= \frac{1}{2}(1 - \xi), \\
\psi_2(\xi) &= \frac{1}{2}(1 + \xi), \\
\psi_3(\xi) &= \frac{1}{4}\sqrt{6}(\xi^2 - 1), \\
\psi_4(\xi) &= \frac{1}{4}\sqrt{10}(\xi^2 - 1)\xi.
\end{align*}
\]

(2.8)  
(2.9)  
(2.10)  
(2.11)

With the use of the shape functions shown above the physical meaning of the internal degrees of freedom can be explained easily. Consider a mono-dimensional element with nodes 1 and 2 at the bar ends and one degree of freedom per node. The nodal displacements are \(u_1\) and \(u_2\). The displacement fields of order \(p\), \(u^{(p)}\) within the bar are

\[
\begin{align*}
u^{(1)}(\xi) &= u_1\psi_1(\xi) + u_2\psi_2(\xi), \\
u^{(p)}(\xi) &= u^{(1)}(\xi) + d_p\psi_p(\xi), \quad p = 2, 3, \ldots, N,
\end{align*}
\]

(2.12)  
(2.13)

where \(d_p\) is the degree of freedom associated to the shape function \(\psi_p\). The corresponding displacement fields are shown in Fig. 2.3. As it can be seen from the pictures, the internal degrees of freedom have no link to the nodal, physical displacements. The generic coefficient \(d_p\) is a measure of the difference between the polynomial approximation of the displacement field of order \(p\) and the one of order \(p - 1\). Again it is clear that enriching the displacement field representation will not need any mesh regeneration, but only the determination of the contribution to the stiffness and mass matrices of the newly added degrees of freedom, bringing a saving in computational costs with respect to the same.

![Displacement fields](image)

Figure 2.3: Displacement fields and their difference from the previous orders for a bar element of order from \(p = 1\) to \(p = 4\) (from [35]).
analysis performed using standard shape functions.

2.3 Hierarchical shape functions of quadrilateral elements

A quadrilateral element contains 4 corner nodes, 4 edges and 1 face. The element is defined in its natural coordinate system $\xi, \eta$ in the range $(-1, +1)$, Fig. 2.4. The shape functions can be divided into three groups corresponding to the element geometry as follows:

![Quadrilateral element](image)

**Figure 2.4:** Quadrilateral element, with the definition of the nodes $N_i$, Edges $E_i$ ($i = 1 : 4$) and polynomial orders $p_\xi, p_\eta$ (from [26]).

1. Nodal shape functions.

   There are 4 bilinear shape functions which are the same as the standard isoparametric four-noded quadrilateral element. The corresponding degrees of freedom are the nodal displacements. The shape function associated to the node $N_i$ is defined by:

   $$\Psi_{1,1}^N(\xi, \eta) = \frac{1}{4}(1 + \xi)(1 + \eta), \quad i = 1 : 4. \quad (2.14)$$

   An example of Nodal shape function is shown in Fig. 2.5.

2. Edge shape functions.

   There are $(p - 1)$ shape functions defined for every edge, vanishing on all other edges. The corresponding degree of freedom is related to the maximum amplitude of the function on the edge on which it is defined. For example the shape function of degree $p$, $(p \geq 2)$ for edge $E_1$, Fig. 2.6 is:

   $$\Psi_{p,1}^{E_1}(\xi, \eta) = \frac{1}{2}(1 - \eta)\phi_p(\xi), \quad (2.15)$$

   where the functions $\phi_p$ are those defined in Eq. 2.6.
3. **Face shape functions.**

There are \((p - 2)(p - 3)/2\) face shape functions, which vanish on each edge, and therefore they are also called *internal* shape functions. For each shape function the associated degree of freedom is related again to the maximum amplitude of the function. For example the internal shape function of degree \(p\) is defined by:

\[
\Psi^\text{int}_{i,j}(\xi, \eta) = \phi_i(\xi)\phi_j(\eta), \quad i + j = p. \tag{2.16}
\]

An example of Face shape function is shown in Fig. 2.7.

### 2.4 Hierarchical shape functions of hexahedral elements

A hexahedral element contains 8 corner nodes, 6 faces and 1 body. The element is defined in his natural coordinate system \(\xi, \eta, \zeta\) in the range \((-1, +1)\), Fig. 2.8.

All shape functions are derived from the product of the mono-dimensional shape functions and can be divided into four groups:

1. **Nodal shape functions.**

   There are 8 trilinear shape functions which are the same as the standard
isoparametric eight-noded hexahedral element. The corresponding degrees of freedom are the nodal displacements. The shape function associated to the node $N_i$ is defined by:

$$
\psi_{1,1,1}^N(\xi, \eta, \zeta) = \frac{1}{8}(1 + \xi)(1 + \eta)(1 + \zeta), \quad i = 1 : 8. \quad (2.17)
$$

2. **Edge shape functions.**

There are $12(p - 1)$ shape functions defined for every edge, vanishing on all other edges. For example the shape function of degree $p, (p \geq 2)$ for edge $E_1$ is:

$$
\psi_{p,1,1}^E(\xi, \eta, \zeta) = \frac{1}{4}(1 - \eta)(1 - \zeta)\phi_p(\xi). \quad (2.18)
$$

![Hexahedral element, with the definition of the nodes $N_i, i = 1..8$, Edges $E_i, i = 1 : 12$ and faces $F_i, i = 1 : 6$ (from [24]).](image)
3. **Face shape functions.**
There are \((p - 2)(p - 3)/2\) face shape functions \((p \geq 4)\), which vanish on each edge. For example the face shape function of degree \(p\), on the face \(F_1\) of Fig. 2, is defined by:

\[
\Psi_{i,j,1}(\xi, \eta, \zeta) = \frac{1}{2}(1 - \zeta)\phi_i(\xi)\phi_j(\eta), \quad i + j = p.
\] (2.19)

4. **Body shape functions.**
There are \((p - 3)(p - 4)(p - 5)/6\) body shape functions \((p \geq 6)\), which vanish on each face. For example the internal shape function of degree \(p\) is defined by:

\[
\Psi_{i,j,k}^{\text{int}}(\xi, \eta, \zeta) = \phi_i(\xi)\phi_j(\eta)\phi_j(\zeta), \quad i + j + k = p.
\] (2.20)

### 2.5 Nastran implementation of p-elements

The commercial finite element package Nastran \(^{37}\) offers the possibility to use the p-elements in performing both static and normal modes analyses. Convergence of a solution can be achieved by reducing the elements size (referred as \(h\)-convergence), or increasing the polynomial order of the shape functions (referred as \(p\)-convergence).

Adaptive analysis can be performed by varying the polynomial order of the shape functions independently in each element, and for each element varying the order along each of the three directions of its natural coordinate system independently. The user can select for each finite element or for a group of finite elements a minimum and a maximum polynomial order for the shape functions for any of the directions of the natural coordinate system. A first solution is calculated using the minimum polynomial order and then the error of the solution is estimated using stress discontinuity and strain energy sensitivity methods \(^{38},^{33}\).

Depending on the error estimation, the polynomial order is increased independently on each local direction, and for each element autonomously. A new solution is calculated and a new error is estimated. The process stops when the error for each element is below a threshold value input by the user or when a maximum number of computational steps has been reached. The adaptive analysis has the advantage of being fully automated, as only one mesh is needed and the changing of the polynomial orders is performed internally, offering the possibility to automate the analysis process.

### 2.6 Modal analysis of flat plates

The determination of the modes and the frequencies of a structure requires the solution of the Eq. 1.6, which in frequency domain can be written in the form:

\[-\omega^2\mathbf{M}\ddot{\mathbf{u}} + \mathbf{K}\ddot{\mathbf{u}} = 0\] (2.21)
2.6. Modal analysis of flat plates

known as the *generalized eigenvalue problem*, where \( \tilde{\mathbf{u}} \) is the Fourier transform of the displacement vector \( \mathbf{u} = \mathbf{u}(\mathbf{x}, t) \). In order to keep the problem as general as possible, it is convenient to make the system non-dimensional. Defining \( L \) a representative dimension of the model’s geometry and in the hypothesis that the material is isotropic, the Young’s modulus \( E \) and the density \( \rho \) are used for the non-dimensionalisation of the equations. The system (2.21) can be rewritten in the following form [39]:

\[
-\hat{\omega}^2 \hat{M} \tilde{\mathbf{v}} + \hat{K} \tilde{\mathbf{v}} = 0 \quad (2.22)
\]

where

\[
\hat{\mathbf{v}} = \frac{\tilde{\mathbf{u}}}{L}, \quad \hat{\omega} = \frac{\omega \rho}{E}, \quad \hat{M} = \frac{M}{\rho}, \quad \hat{K} = \frac{K}{E} \quad (2.23)
\]

The non-dimensionalisation has been performed in Nastran setting \( E = 10^6 \) (this high value has been chosen to avoid numerical problems with too small frequencies), \( \rho = 1 \) and the Poisson number to \( \nu = 0.225 \), equal to the one of Aluminum.

In this section a modal analysis of an unconstrained flat, squared plate of isotropic material having different aspect ratios (ratio between the side length of the square plate and the thickness) is performed. The plate is modeled using solid “brick” elements with p-formulation (CHEXA) and four noded linear shell elements (CQUAD4). Both finite element models use a consistent formulation for the mass matrix, defined by Eq. (1.7). The convergence rates of the natural frequencies versus the number of degrees of freedom are evaluated. Results are presented up to the fourth non-rigid mode. The modes are named according to the corresponding frequency, in ascending order, so the first mode corresponds to the lowest (non-zero) frequency. The frequencies results are truncated to the third significative digit. From a purely mathematical standpoint, all significative digits should be retained in the presentation of the results but, for the purposes of a preliminary design environment, few digits can be acceptable for evaluating the convergence of a computation.

![Figure 2.9: Flat plate geometrical parameters.](image)

2.6.1 Thick plate

The plate analysed has dimensions of \( L_1 = 1, L_2 = 1 \) and \( L_3 = 0.01 \). According to the classical plate theory [24] a plate with such a thickness is referred as a “thin” plate, with all the deductions coming from such hypothesis. Differently
from the classical theory this plate is referred as thick as its thickness is of the order of the centimeter, thus one order of magnitude higher than the thickness of typical aeronautical panel, whose thickness is of the order of the millimeters.

When the plate is meshed using solids the number of elements along the thickness must be specified. As the thickness is “small” compared with the in-plane dimensions of the plate, this should imply that less elements are needed along the thickness. To investigate this aspect a sensitivity analysis has been done on the effect of the number of elements chosen for the mesh along the thickness on the calculated natural frequencies. For this sensitivity analysis the polynomial order of the p-elements has been kept fixed and the convergence has been checked just increasing the number of elements in the in-plane directions, keeping fixed the number of elements along the thickness. Figure 2.10 shows the trend of the second and third frequencies vs the number of degrees of freedom for the three models obtained by meshing the plate using one, two and three elements along the thickness. As can be seen, for each solution point computed, the curves corresponding to an increasing number of thickness-wise elements are shifted towards a higher number of degrees of freedom. Therefore it can be concluded that the solution does not depend on the number of elements along the thickness; thus, in the following calculation only one element will be put along the thickness of the plate.

In addition, the p-formulation allows the choice of the polynomial order of the shape functions along the three edges of a solid element. Because of the small thickness of the plate, the solution is expected not to be sensitive to the choice of the polynomial order along the thickness provided that it is high enough to prevent locking of the element. Thus, a sensitivity analysis on the choice of the order of the polynomials along the element thickness has been performed.

Figure 2.11 shows the trend of the second and third frequencies vs the number of degrees of freedom of the system considering three different choices for the shape functions order: fifth order for the two in plane directions and fifth, third, and second order along the thickness, labeled respectively brick-p5, brick-p553, and brick-p552. The figure shows that when the order of the polynomials along
2.6. Modal analysis of flat plates

Figure 2.11: Sensitivity analysis for the polynomial order choice along the thickness. Thick plate, $L_1 = 1, L_2 = 1, L_3 = 0.01$; one element along the thickness.

The thickness is lowered, the solution is not affected at all, that is, the curves are shifted on the left-hand side, which indicates reduction of the size of the system, thus, a reduction in the computational time.

Following these results, the calculations performed with the solid p-elements have been done with one element and second-order polynomials along the thickness. A convergence analysis of the natural frequencies of the squared plate has been performed for both shell and solid models, by increasing the number of elements along the in-plane edges, i.e. in the standard way (h-convergence). Mode shapes of the first four modes are shown in Fig. 2.12. For the solid model, based on p-elements, the choice of the polynomial order is free.

Three different orders have been selected for the in-plane edges, in particular, third, fifth and seventh, resulting in three solid models. These models are referred in the pictures as brick-p332, brick-p552, brick-p772.

Figure 2.13 represents the frequencies versus the number of degrees of freedom. The meshes selected start from one element per edge up to the number of elements at which convergence is achieved. The result is considered converged when the last three runs give the same result (first three significative digits coincident) for the frequencies. In Fig. 2.13 the result obtained using one element per edge for the case brick-p332 has been removed as it was off more than 20% from the converged result. Figure 2.13 shows that the solid model is able to behave exactly as the shell model as converged values are the same except for the first,
pure shear mode, for which the shell model predicts a converged value of 6.63 Hz whereas the solid models give a value of 6.64 Hz. For the shear dominated modes, Figs. 2.13a, 2.13d, the convergence rate of the solid and shell models are almost the same. For the bending dominated modes Figs. 2.13b, 2.13c, the p-version has a rate of convergence higher than the shell model which also exhibits a non-monotonic trend due to the very coarse mesh used. Increasing the polynomial orders for the solid, the convergence rate of the increases, as expected. In particular, using a seventh order polynomial expansion the converged results can be obtained just using one element to mesh the square plate.

The traditional approach for the convergence analysis requires the generation of several meshes. Using the p-elements another approach can be followed: generating one single mesh and increasing the polynomial order until a converged result has been achieved. A comparison between the standard and the polynomial convergence analysis is performed. Figure 2.14 shows the convergence curves for the shell model and the solid model, obtained increasing the number of elements. In addition the convergence curves for the solid model, performing an adaptive analysis are plotted: in this case an initial mesh is chosen, and then starting from
an third polynomial expansion, the order of the polynomials is increased until the seventh order is reached. Three initial different meshes are chosen, based on one, two and three elements per (in-plane) edge. The corresponding convergence curves are named respectively brick-p772-adapt-1, brick-p772-adapt-2, brick-p772-adapt-3. The maximum polynomial order is set to the seventh order, showing that the final result is coincident with the result obtained in the standard way, choosing a mesh and a fixed polynomial order (curve named brick-p772). Of course it can be raised to any order until convergence is achieved.

The pictures show that for the first three modes, using just one finite element, a converged result can be obtained with at least the same rate of convergence as the shell model, but using less calculation steps (only three) and with less degrees of freedom. For the fourth mode a converged result can be achieved in three calculation steps, using a mesh of $2 \times 2 \times 1$ elements whereas for the shell model the number of calculation steps (each one associated to a completely new mesh generation) is at least six. For better understanding the pictures are zoomed around the points of the adaptive analysis, leaving out most of the points of the shell convergence analysis.
2.6.2 Thin plate

The plate analysed has dimensions of $L_1 \times L_2 \times L_3 = 1 \times 1 \times 0.001$. The same analyses carried out for the thick plate are performed for this case. For the solid model only one element will be considered along the thickness and the polynomial order along the thickness can be limited to the second order, as shown in Fig. 2.15. A comparison between the convergence behavior of the shell model and different solid models is performed. The convergence analysis for the shell model is performed increasing the number of elements whereas the analysis for the solid model is performed using an adaptive method. Three different initial meshes are chosen, based on one, two and three elements per edge. The minimum polynomial order chosen is three and the maximum is seven. The curves corresponding to the three meshes are referred in the plots as brick-p772-adapt-1, brick-p772-adapt-2, brick-p772-adapt-3. Besides the adaptive analysis, a standard analysis is performed for the solid model, increasing the number of elements and keeping the polynomial order fixed to seven. The corresponding curve is referred in the plots as brick-p772. Results are reported in Fig. 2.16. The adaptive analyses with solid elements show a convergence rate higher than the shell models for the brick-p772-adapt-2, brick-p772-adapt-3 models, whereas the brick-p772-adapt-1 model has a convergence rate comparable with the shell model which instead needs more degrees of freedom to achieve convergence, and more calculation steps, with the need to regenerate a new mesh for each computation. Besides, each solid model needs at most three computations to achieve convergence, and in some cases the brick-p772-adapt-2, brick-p772-adapt-3 models need only one computation and less degrees of freedom than the shell model.

2.7 Modal analysis of curved panels

In the standard approach of the finite element method the mesh refinement guarantees that the geometry is approximated more and more accurately. In
2.7. Modal analysis of curved panels

![Graphs showing frequency vs degrees of freedom for different modes of a thin plate.](image)

**Figure 2.16:** Comparison of the convergence rate of the shell and solid models for the first four modes of a thin plate.

Particular the shape functions describing the unknown displacement field are also used to describe the geometry and the finite element is referred as *isoparametric*. At the limit, when the reference size of the largest element tends to zero, the representation of the domain is exact. In the p-version of the finite element method, the mesh, rather coarse, is left unchanged and convergence is achieved increasing the order of the polynomial order of the shape functions. Therefore it’s important that the geometry of the structure is accurately modelled with the (little) number of elements used. Different methods have been developed to describe complex geometries with few elements, one of the most popular is the so-called *blending functions method*, from Gordon and Hall [40]. In this case the mapping from the local coordinates to the global, cartesian coordinates of the element is made of the standard (linear) mapping of isoparametric elements and two additional mappings consisting in a face and an edge blending [26]. The additional mappings provide a higher order representation for the geometry of the element.

In Nastran the curved edges of finite elements are described as rational parametric cubic curves [41] and the faces are defined as rational parametric bicubic surfaces [41]. In order to achieve such a description sixteen additional points,
equally spaced in the two parametric coordinates of a surface, must be specified in the input file for each curved surface of the finite elements generated. Some of these points coincide with the corner nodes of the finite element and therefore their positions need not to be determined. An example is given in Fig. 2.17 where the points needed for the description of the higher order geometry of a single curvature panel are reported. As can also be seen from this figure, on each edge four points are used to build a third order polynomial description of the same edge. Therefore $C^1$ continuity is achieved, but this is not assured at the common nodes of two contiguous elements. Nastran solver provides a solution to this problem by moving the two internal points of each edge in order to ensure $C^1$ continuity of adjacent edges for contiguous elements. This is done in an automatic manner, transparent to the user, during the assembling phase of the mass and stiffness matrices.

2.7.1 Thin cylindrical panel

The mean surface of the panel has dimensions of $L_1 \times L_2 \times L_3 = 1 \times 1 \times 0.001$. A finite element model is created using linear shell elements. For the sake of clarity, it must be remarked that although in the Nastran implementation these elements (CQUAD4) are referred as shells, in reality they are plate elements. This means that these elements have no curvature, thus membrane and bending behavior are uncoupled. Conversely in a curved element this coupling is unavoidable. In addition, several elements are needed to approximate accurately the geometry of a curved panel. Nevertheless, these elements will be always referred as shell elements in the following. Different modal analyses are performed generating several meshes. Each new analysis is performed doubling the number of elements. A solid model is then generated. A standard convergence analysis is performed increasing the number of elements along the in-plane edges, whereas along the thickness the number of elements is fixed to one, and the polynomial order is kept constant. An adaptive analysis is then performed considering three different meshes having one, two and three elements per (in-plane) edge.

The polynomial order starts from three up to nine along the in-plane edges

![Cylindrical panel geometrical parameters.](image)

![One element mesh: corner nodes (point circles) and points for the higher order geometry description (crosses).](image)

Figure 2.17: Cylindrical panel geometry and example mesh.
2.7. Modal analysis of curved panels

whereas along the thickness the order is kept to two. The comparison of the different results is reported in Fig. 2.18 where the curves corresponding to the three adaptive analyses are referred as brick-p992-adapt-1, brick-p992-adapt-2, brick-p992-adapt-3. Mode shapes are reported in Fig. 2.19. For the pure shear mode, Figs. 2.18a and 2.19a, the shell model reach a converged value with less degrees of freedom, whereas for the other modes the solid models exhibit a higher convergence rate and attain a converged value with less degrees of freedom. The higher convergence rate is exhibited by the bending dominated modes, Figs. 2.18b, 2.18d and Figs. 2.19b, 2.19d. For the adaptive analysis made with solids, two or three computations are sufficient to get a converged results, whereas for the shell model, the number of computations is almost double. In addition, for the last three modes the convergence history of the shell model is not monotonic, due to both mesh coarseness and the inability of the Nastran shell elements to capture the curved geometry of the panel as they are in reality flat (plate) elements.

Figure 2.18: Comparison of the convergence rate of the shell and solid models for the first four modes of a thin cylindrical panel.
2.7.2 Thin spherical panel

The mean surface of the panel analysed has dimensions of $L_1 = 1$, $L_2 = 1$, $L_3 = 0.001$, Fig. 2.20a. A comparison between the convergence behavior of the shell model and different solid models is performed. The convergence analysis for the shell model is performed doubling the number of elements for each run whereas the analysis for the solid model is performed using an adaptive method with a constant number of elements. Three different initial meshes are chosen, based on two, three and four elements per edge. The minimum polynomial order chosen is three and the maximum is twelve, as due to the double curvature the convergence rate is expected to be slower. One element is taken along the thickness.

The curves corresponding to the three solid meshes are referred into the plots as brick-p12122-adapt-2, brick-p12122-adapt-3, brick-p12122-adapt-4. Besides the adaptive analysis, a standard analysis is performed for the solid model, increasing the number of elements and keeping the polynomial order fixed to twelve. The corresponding curve is referred in the plots as brick-p12122.

Results are reported in Fig. 2.22. The adaptive analyses with solid elements exhibit a convergence rate higher than the shell models, but in this case the converged value is slightly different depending on the initial mesh used, so in
2.7. Modal analysis of curved panels

(a) First mode  (b) Second mode  (c) Third mode  (d) Fourth mode

Figure 2.21: First four mode shapes of the spherical panel.

In this case the solid formulation needs elements with lower aspect ratio than a single curvature or a flat panel. A geometrical description better than the cubic and bi-cubic representation used in the current Nastran implementation of the p-elements would certainly help in case of double or high curvature structural elements. The converged value is achieved using about the same number of degrees of freedom of the shell model, except for the first, pure shear mode, where the shell model performs better than the solid one. In any case the number of calculation steps required by the shell model is almost double the number of...

Figure 2.22: Comparison of the convergence rate of the shell and solid models for the first four modes of a thin spherical panel.
computations for the solid model, and for each computation a new mesh must be generated. This is easy for a simple panel, it’s practically not affordable for a more complex model, like a wing structure. It must also be noted that the convergence curves of the solid models are always monotonic, and therefore extrapolation techniques can be used to predict the converged frequency value, whereas this is not always possible for the shell models. Indeed the third mode convergence history exhibits a non-monotonic behavior that can be imputed to the fact that the Nastran shells are in reality flat plates. This could also explain why the convergence of the last two modes is achieved from a frequency lower than the asymptotic value, whereas it is expected that coarse models are always “stiffer” than very fine models.

2.7.3 Mass matrix formulation

For all the comparisons shown so far, the solid and shell models use a consistent mass matrix formulation, which results in a full mass matrix. In the industrial practice instead, a lumped mass matrix formulation is employed for the shell models, resulting in a diagonal mass matrix. This eases the distribution of the non-structural masses over e.g. a wing structure as the non-structural masses are defined as point masses concentrated in certain nodes. The diagonal structure of the mass matrix prevents that the masses are spreaded over nodes different from the ones they are attached to.

The p-formulation of the finite elements instead uses a consistent mass matrix formulation. The mass matrix is full and in this case it’s difficult to evaluate the cross coupling defined by a concentrated mass. Using a full mass matrix makes troublesome the use of time domain techniques to perform the explicit integration of the equation of motion, as it is done for example when the structural operator in non-linear. Nevertheless, when a structure is modeled using solid p-elements, the non-structural masses can be taken into account easily as it will be shown in Chapter 5.

The choice between a consistent or a lumped mass matrix for a shell model has also an effect on the convergence rate of the natural frequencies. Figure 2.23 reports the convergence analysis for the double curvature panel analysed before. A shell model based on both lumped and consistent mass matrices is created as well as a solid model. Convergence is achieved doubling the number of elements for the shell model, whereas for the solid model an adaptive analysis is performed, based on a fixed mesh with three elements per in-plane edge and one along the thickness.

The figure shows that the lumped mass model requires more computations to achieve convergence than with the consistent mass model. Besides, for a fixed number of degrees of freedom the results can be much farther from the converged solution than the consistent mass model. The solid model shows its better behavior as few computations are needed to achieve convergence.
2.8 Discussion

The results presented in this chapter show how solid, p-version finite elements, can be efficiently used to model different kinds of panels, obtaining the same results as with a traditional shell linear model, without any problem in modelling and analysing very thin structural elements.

The solid p-elements can solve the problems connected with the consistency and the convergence of a finite element model whenever an automated modeling and analysis approach is taken to support a KBE based design process: different kinds of panels can be modeled using the same type of finite element, independently by the nature of the structural element, thin or thick walled; in addition, the convergence checks can be performed by running an adaptive analysis which gives the possibility to use just one mesh without regenerating a new finite element model after a solution has been computed.

The solid modelling approach has been validated against shell models for the different panels considered. In all of the cases examined the solid model attains a converged result using less computational steps (i.e. number of adaptive calculations for the solids, number of mesh refinements for the shells) and in most cases.

Figure 2.23: Comparison of the convergence rate of the shell and solid models for the first four modes of a thin spherical panel.
of the cases using less degrees of freedom.

The fact that the shell model sometimes attains a converged result for the shear dominated modes faster than the solid model is probably due to the fact that the shell model contains as degrees of freedom the rotations, whereas the solid model has only the translations, and the rotations are derived quantities, so less accurate.

It must be remarked that the efficiency of the two methods is best measured by the time needed to obtain a converged result. For this aim the number of computations and the number of degrees of freedom are not more than an indication, as a big difference between a shell and a solid p-element model is in the structure of the stiffness and mass matrices. For a shell model the stiffness and mass matrices are sparse, whereas a solid, p-formulation model has full stiffness and mass matrices. The bandwidth of these matrices increases with the polynomial order chosen. Given two finite element models having the same number of degrees of freedom, one made of shells and the other one made of solid p-elements, the CPU time required to compute a solution (frequencies and mode shapes) is for the solid model higher than the shell model. The CPU time of a solid p-elements model increases rapidly with the number of degrees of freedom, ranging from few seconds for the lower order models shown in the preceding sections to tenths of seconds for the higher polynomial orders (ninth for example). Although important, this kind of comparison is not completely meaningful. Indeed the comparison should be done in terms of the time needed to achieve a converged solution. This time, in the case of shell models, must include also the time needed to (re-)generate the mesh during the convergence analysis process. Clearly if the mesh (re-)generation is done by hand, as it has been done for the different panel models shown so far, the solid p-element models are outperforming the shell models because the p-elements do not require any mesh re-generation. However, even if an automatic procedure is available for the generation of finer shell meshes, the results obtained have shown that a solid, p-formulation finite element model:

- requires less computational steps than a shell model;
- requires less degrees of freedom than a shell model;
- for each computational step it does not require any remeshing. Just the contribution of the newly added degrees of freedom must be added to the existing matrices;
- can efficiently exploit static reduction techniques as the degrees of freedom associated to the body shape functions are purely local to the element, thus reducing further the number of degrees of freedom;
- give even better results if the same computations for the shell model are performed using a lumped mass matrix, as commonly done in the industrial practice.

A shell model, instead, requires a new mesh generation for each new calculation, thus a new preprocessing step, which can be easily automated for a simple panel, but is not affordable for a more complex structure, such as an aircraft.
Chapter 3

Solid P-Formulation Finite Elements for Composite Structures

3.1 Introduction

The capabilities offered by the solid p-elements in performing the modal analysis of thin-walled panels made of isotropic materials are now going to be tested on the modal behavior of panels made of composite materials.

New aircraft generations have an ever increasing percentage of composite materials, build-up from unidirectional laminae or fabrics. Appropriate tools to support the design and analysis of composite structures in the preliminary design stage are therefore becoming more important. In the following sections a method for the modelling of a composite structure using solid p-elements will be defined and tested using the p-elements available in the commercial FE package Nastran[^2].

3.2 Solid orthotropic model for composite plates

A unidirectionally reinforced or a woven lamina, can be reasonably considered as an orthotropic sheet of material that, when loaded, shows an internal plane stress state[^3]. The elastic behavior of the lamina can then be defined by a reduced stiffness matrix or a reduced compliance matrix, the latter having the closest relation to the engineering constants of the material. The reduced stiffness matrix is defined by the following stress-strain relations, in the material principal frame of reference, Eq. (3.1) and in a general frame of reference, Eq. (3.2):

\[
\begin{bmatrix}
\sigma_1 \\
\sigma_2 \\
\tau_{12}
\end{bmatrix} =
\begin{bmatrix}
Q_{11} & Q_{12} & 0 \\
Q_{12} & Q_{22} & 0 \\
0 & 0 & Q_{66}
\end{bmatrix}
\begin{bmatrix}
\epsilon_1 \\
\epsilon_2 \\
\gamma_{12}
\end{bmatrix},
\]  

(3.1)

\[
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix} =
\begin{bmatrix}
Q_{11} & Q_{12} & Q_{16} \\
Q_{12} & Q_{22} & Q_{26} \\
Q_{16} & Q_{26} & Q_{66}
\end{bmatrix}
\begin{bmatrix}
\epsilon_x \\
\epsilon_y \\
\gamma_{xy}
\end{bmatrix}.
\]  

(3.2)
From the laminate theory, the relations between force and deformations of a thin laminate made of a stack of N laminae having different orientations are:

\[
\begin{bmatrix}
N_x \\
N_y \\
N_{xy}
\end{bmatrix} = \begin{bmatrix}
A_{11} & A_{12} & A_{16} \\
A_{12} & A_{22} & A_{26} \\
A_{16} & A_{26} & A_{66}
\end{bmatrix} \begin{bmatrix}
\epsilon_x^0 \\
\epsilon_y^0 \\
\epsilon_{xy}^0
\end{bmatrix} + \begin{bmatrix}
B_{11} & B_{12} & B_{16} \\
B_{12} & B_{22} & B_{26} \\
B_{16} & B_{26} & B_{66}
\end{bmatrix} \begin{bmatrix}
k_x \\
k_y \\
k_{xy}
\end{bmatrix},
\]

(3.3)

\[
\begin{bmatrix}
M_x \\
M_y \\
M_{xy}
\end{bmatrix} = \begin{bmatrix}
B_{11} & B_{12} & B_{16} \\
B_{12} & B_{22} & B_{26} \\
B_{16} & B_{26} & B_{66}
\end{bmatrix} \begin{bmatrix}
\epsilon_x^0 \\
\epsilon_y^0 \\
\epsilon_{xy}^0
\end{bmatrix} + \begin{bmatrix}
D_{11} & D_{12} & D_{16} \\
D_{12} & D_{22} & D_{26} \\
D_{16} & D_{26} & D_{66}
\end{bmatrix} \begin{bmatrix}
k_x \\
k_y \\
k_{xy}
\end{bmatrix},
\]

(3.4)

where the terms of the \( A \), \( B \) and \( D \) matrices are defined as:

\[
A_{ij} = \sum_{k=1}^{N} (Q_{ij})_k (z_k - z_{k-1}),
\]

(3.5)

\[
B_{ij} = \frac{1}{2} \sum_{k=1}^{N} (Q_{ij})_k (z_k^2 - z_{k-1}^2),
\]

(3.6)

\[
D_{ij} = \frac{1}{3} \sum_{k=1}^{N} (Q_{ij})_k (z_k^3 - z_{k-1}^3).
\]

(3.7)

The above mathematical representation is the result of the assumption that the in-plane displacement field components (in-plane with respect to the laminate plane, for a flat laminate) have a linear variation along the thickness of the laminate. This assumption eventually eliminates the dependence of the displacements from the thicknesswise coordinate of the laminate. But in our case the solid modelling reintroduces this dependence. We could then follow two approaches to tackle the problem: modeling each lamina of the laminate as a solid, but this is too computationally expensive for our goals, or better, consider the laminate as made of a general anisotropic and homogeneous material. Its properties can be derived either by experimental tests or, as done in this chapter, just using the Eqs. 3.5-3.7 in a backward manner as from the components of the matrices \( A \), \( B \), \( D \), using Eq. 3.8 we can calculate back the stiffness matrix \( Q \) for the solid model. In particular, in the hypothesis that the laminate consists of only one-layer of homogeneous and orthotropic material it comes out that:

\[
A_{ij} = Q_{ij}^l, \quad B_{ij} = 0, \quad D_{ij} = \frac{Q_{ij}^l l^3}{12}.
\]

(3.8)

From Eq. 3.8 it can be seen that for the determination of the components of the matrix \( Q \) we have two equations in one unknown. As a consequence, it should be decided from which equation determine the \( Q_{ij} \) values, thus choosing if the solid model should match the membrane/shear (\( A \) matrix) or the bending/torsion (\( B \) matrix) behavior. In any case the behavior of the one-layer solid laminate and the layered laminate will be different. For a wing-skin or a fuselage panel a reasonable choice is to derive the \( Q_{ij} \) values from the \( A \) matrix, as they carry mainly membrane and shear loads. From the \( Q_{ij} \) values the four independent
3.3. Applications

Engineering constants $E_1$, $E_2$, $G_{12}$ and $\nu_{12}$ can be determined, using the following relations:

\[
Q_{11} = \frac{E_1}{1 - \nu_{12}\nu_{21}}, \quad Q_{22} = \frac{E_2}{1 - \nu_{12}\nu_{21}}, \quad \frac{\nu_{12}}{E_1} = \frac{\nu_{21}}{E_2}, \quad Q_{66} = G_{12}. \tag{3.9}
\]

\[
Q_{12} = \frac{\nu_{12}E_2}{1 - \nu_{12}\nu_{21}} = \frac{\nu_{21}E_1}{1 - \nu_{12}\nu_{21}}, \quad Q_{66} = G_{12}. \tag{3.10}
\]

As the laminate is modeled as a solid we will need the other five missing constants, $E_3$, $G_{13}$, $G_{23}$, $\nu_{13}$ and $\nu_{23}$ which can be used to derive the flexibility matrix for the one-layered laminate. The flexibility matrix $S$ appears as:

\[
[S_{ij}] = \begin{bmatrix}
\frac{1}{E_1} & -\frac{\nu_{21}}{E_2} & -\frac{\nu_{31}}{E_3} & 0 & 0 & 0 \\
-\frac{\nu_{12}}{E_1} & \frac{1}{E_2} & -\frac{\nu_{32}}{E_3} & 0 & 0 & 0 \\
-\frac{\nu_{13}}{E_1} & -\frac{\nu_{23}}{E_2} & \frac{1}{E_3} & 0 & 0 & 0 \\
0 & 0 & 0 & 1/G_{23} & 0 & 0 \\
0 & 0 & 0 & 0 & 1/G_{23} & 0 \\
0 & 0 & 0 & 0 & 0 & 1/G_{12}
\end{bmatrix}. \tag{3.11}
\]

The determination of the five missing constants, describing the off-plane elastic properties, can be done through physical considerations:

- in the case of a unidirectional lamina, the $E_{33}$ value is determined by the elasticity of the resin, so it equal to the $E_2$ value of the lamina itself;
- the $\nu_{23}$ and $\nu_{31}$ values have been made equal to the $\nu_{12}$ value as a sensitivity analysis can show that the Poisson values have almost no influence on the modal frequencies;
- the $G_{13}$ and $G_{23}$ values have been set equal to the $G_{12}$ value. For thin walled structural elements it is not important to have the exact value for these constants as they behave as shear indeformable elements, so the only important thing is not to give values of zero.

3.3 Applications

The equivalencing method is tested on the flat sandwich panel which has been extensively studied in [43]. Indeed the dynamic behavior of sandwich panels is determined by the in-plane extensional and shear stiffness of the facings just like the behavior of the wing box is determined by the extensional and in-plane shear stiffness of skin panels and spars.

The sandwich panel is made of composite facings and a titanium honeycomb core. It has planar dimensions of $90 \times 90$ cm and a $2.54$ cm core thickness. Geometrical and elastic properties for the components are taken from [43] and reported in Tables 3.1 and 3.2. Lamination sequence is $[45, -45, 90, 0]_s$. In the reference no data are given for the Poisson’s coefficients of the honeycomb core, so zero values have been assigned to the three Poisson values as they can be either positive or negative. Only the $\nu_{13} = 0.34$ value has been set equal to the value of the Titanium alloy used to make the honeycomb, as stated in [44].
Table 3.1: Material properties of the Titanium honeycomb core (from [43]).

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weight Density</td>
<td>$\rho (N/m^3)$</td>
</tr>
<tr>
<td>Young’s Modulus $E_{11}$</td>
<td>1066.5</td>
</tr>
<tr>
<td>Young’s Modulus $E_{22}$</td>
<td>1.65E+05</td>
</tr>
<tr>
<td>Young’s Modulus $E_{33}$</td>
<td>1.03E+09</td>
</tr>
<tr>
<td>Shear Modulus $G_{13}$</td>
<td>4.82E+08</td>
</tr>
<tr>
<td>Shear Modulus $G_{23}$</td>
<td>4.82E+08</td>
</tr>
<tr>
<td>Shear Modulus $G_{12}$</td>
<td>3.38E+08</td>
</tr>
</tbody>
</table>

In the following section 3.3.1, finite element models will be discussed for a composite plate having the same elastic properties as the facings of the sandwich panel. These models will be used to show some obvious but important consequences of the assumptions made for the equivalent model, and to show some guidelines in the finite element modelling and analysis using solid p-elements.

In section 3.3.2 a convergence analysis of the modal frequencies of the sandwich panel will be done comparing the different kinds of equivalent and original models.

3.3.1 Unconstrained Facing Analysis

The technique for building equivalent models of laminates is tested first on a composite plate, having the properties of the facings of the sandwich panel. The modal analysis is carried out considering three different finite element models: the first, model (1) (see Fig. 3.1), regarded as reference, made of 4-node plate linear elements (CQUAD4) whose elastic properties are specified assigning each ply and its orientation through a PCOMP card in Nastran, the second, model (2) (see Fig. 3.2a) made again of CQUAD4 elements whose properties are assigned considering the plate as a single layered orthotropic material and the third model (3) (see Fig. 3.2b) made of 8-node brick p-elements (CHEXA, p-formulation) whose elastic properties are again assigned considering it as a single layered orthotropic material. In the latter two cases the engineering constants of the orthotropic materials are determined using the equivalencing procedure described in section 3.2.

Table 3.2: Material Properties of the Im7/Peti5 Layers used for the facings (from [43]).

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s Modulus $E_{11}$</td>
<td>151.92</td>
</tr>
<tr>
<td>Young’s Modulus $E_{22}$</td>
<td>9.646</td>
</tr>
<tr>
<td>Shear Modulus $G_{12}$</td>
<td>2.564</td>
</tr>
<tr>
<td>Poisson’s Ratio $\nu_{12}$</td>
<td>0.34</td>
</tr>
<tr>
<td>Weight Density $\rho (N/m^3)$</td>
<td>15750</td>
</tr>
<tr>
<td>Ply Thickness $t (m)$</td>
<td>1.397E-04</td>
</tr>
</tbody>
</table>
3.3. Applications

The basic assumption of matching only the membrane and shear behavior of the facing generates differences between the equivalent and the reference model. As an example in Table 3.3 are reported the frequencies determined for the three models, for a 12x12 meshing (considered in [43] as converged values). As can be seen there are large differences between the model (1) and the models (2) and (3). It is clear also that there’s almost no difference in the values of the frequencies predicted by the equivalent plate and solid finite-element models, (2) and (3). This just confirms the ability of the solids to behave as thin plate elements. When the facing is meshed with solids, the number of elements along the thickness must be specified. The “small” thickness of the facing should infer that the results do not depend on the number of elements. For this reason a sensitivity analysis has been done, and in Fig. 3.3a-3.3b are reported the trends of the first two frequencies versus the number of degrees of freedom for 1, 2 and 3 elements along the thickness. As can be seen the solution doesn’t depend at all on this number, thus in the following calculation only 1 element will be put along the thickness of the panel.

Besides, p-formulation allows the choice of the polynomial order of the shape functions along the three edges of a solid element. Due to the small thickness of
the plate, the solution is not sensitive to the choice of the polynomial order along
the thickness provided that it is high enough to prevent locking of the element.
Thus a sensitivity analysis on the choice of the order of the polynomials along
the element thickness has been carried out. In Fig. 3.4a-3.4b are reported the
trends of the first four frequencies versus the number of degrees of freedom of
the system, for the single layered solid model (3), considering three different
choices for the shape functions order: fifth order for the two in plane directions
and fifth, third and second-order along the thickness, labeled respectively brick-p5,
brick-p5p3 and brick-p5p2. The charts show that lowering the order of the
polynomials along the thickness the solution is not affected at all, ie. the curves
are shifted on the left hand of the picture, meaning a reduction of the size of the
system thus a reduction in the computational time. Following these results, the
calculations using solid p-elements have been done with one element and second
order polynomials along the thickness.

### 3.3.2 Unconstrained flat sandwich panel

The technique for building equivalent models of laminates is finally tested on the
flat sandwich reported in [43]. As for the facing analysis, three different models
have been considered; all of them have the core modeled using solid 8-node finite
elements (CHEXA) and the facings modeled in three different ways: the first

![Graph 1](image1)
![Graph 2](image2)

**Figure 3.3:** Effect of the number of elements choice along the thickness of the unconstrained facing on the convergence analysis of the first two lower frequencies versus the number of degrees of freedom.
3.3. Applications

Figure 3.4: Effect of the shape functions polynomial order choice along the thickness-wise edges of the unconstrained facing on the convergence analysis of the first two lower frequencies versus the number of degrees of freedom.

The first frequency.

The second frequency.

Model (1) (see Fig. 3.1), considered as reference, is made of 4-node linear elements (CQUAD4) whose elastic properties are specified assigning each individual ply and its orientation through an PCOMP card in Nastran, the second (2) and the third (3) models (see Figs. 3.2a-3.2b), made respectively of 4-node linear plate (CQUAD4) and 8-node brick p-elements (CHEXA, p-formulation) whose elastic properties are determined using the equivalencing method introduced in section 5.2.

Model (3) is thus entirely made of solid finite elements, but the number of the elements along the thickness is kept equal to three (one for the upper facing, one for the core and one for the lower facing), and the polynomial order of the shape functions along the thickness is kept equal to 2, eliminating any dependence of the model behavior on the thickness. As the p-elements offer the possibility to run an analysis changing the polynomial orders of the shape functions without changing the mesh, three different full solid models (Model (3) type) have been considered, corresponding respectively to a one finite-element, two finite-elements and three finite-elements mesh along the in-plane edges of the sandwich plate.

A convergence analysis of the modal frequencies of the sandwich plate has been performed for each of the models considered, by increasing the number of elements along the in-plane edges for the shell based models and increasing the order of the shape functions for the full solid models. The (in-plane) polynomial orders start from the second order and they increased up to order $p = 12$ for the coarsest solid model, whereas for the other solid models they are increased up to order $p = 9$. The mode shapes of the first four modes for the model (3) (they are the same for the other models) are reported in Fig. 3.5. Table 3.4 reports the converged values for the different models, where the label shell-laminated refers to model (1), label shell-equivalent refers to model (2), and labels brick-1-element, brick-2-elements, brick-3-elements, refer to the one-element, two-elements and three-elements in-plane meshes for the model (3).
As can be seen all the models give the same value (up to the third digit): the differences among the reference model and the several equivalent models have disappeared. This confirms the validity of the equivalencing method introduced above at a wing-box level: it is sufficient to match only the membrane and shear behavior of the full model.

The trend of the different models towards the converged values are presented in Figs. 3.6a-3.6d. On the x-axis are reported the number of degrees of freedom and on the y-axis are reported the calculated frequencies. As can be seen some points fall outside the range chosen in the figures. These points correspond to the coarsest meshes results for the shell models and to the minimum shape functions order (second) results for the solid models. The pictures show that the shell models have an oscillating behavior such that a converged value is not reached even with very fine meshes (the last runs for the models (1) and (2) are done using 150 elements in the in-plane edges). This behavior could be explained with the fact that the models (1) and (2) use a mix of solid and shell elements. Whereas solid elements have only translational degrees of freedom, the shell elements have also rotational degrees of freedom, which are not transferred to the solid models. Keeping this degree of freedom unconstrained could have an influence on the convergence history of the two mix models. This could be a reasonable explanation as such oscillations are not exhibited by the convergence histories of the shell models analysed in Chap. 2. Those models are made by the same type of shell element (CQUAD4) and use the same type of eigenvalue problem solver (Lanczos [37]), with the same analysis set-up. Differently from the shell models, the solid models instead reach a converged value using much less degrees of freedom (almost two order of magnitudes reduction) for any of the three solid meshes considered. Looking at the asymptotic behavior, the results

<table>
<thead>
<tr>
<th>Mode</th>
<th>Shell laminated</th>
<th>Shell equivalent</th>
<th>Brick 1-element</th>
<th>Brick 2-elements</th>
<th>Brick 3-elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>1st</td>
<td>153.14</td>
<td>153.16</td>
<td>153.19</td>
<td>153.19</td>
<td>153.19</td>
</tr>
<tr>
<td>2nd</td>
<td>224.67</td>
<td>224.70</td>
<td>224.71</td>
<td>224.71</td>
<td>224.71</td>
</tr>
<tr>
<td>3rd</td>
<td>285.32</td>
<td>285.14</td>
<td>285.16</td>
<td>285.16</td>
<td>285.16</td>
</tr>
<tr>
<td>4th</td>
<td>388.64</td>
<td>388.64</td>
<td>388.63</td>
<td>388.63</td>
<td>388.62</td>
</tr>
</tbody>
</table>
show that the solid p-elements are outperforming over the shell models. But it must be remarked that from an engineering point of view, in a preliminary design environment the goal is to achieve a “sufficiently” accurate solution using a limited amount of resources, and therefore a few digits accurate result can be considered satisfactory in a preliminary design analysis.

For this reason Fig. 3.7 reports the convergence analysis in terms of the relative error versus the number of degrees of freedom. For each of the five models considered in Fig. 3.6 the relative error is computed as the normalised difference between the converged frequency and the frequency computed for any of the test runs. The normalisation is done with respect to the converged frequency for the model considered.

The scale of the relative error has been limited to 10% in the pictures, cutting out the first computation for each of the models. For the solid model, one or two computations are sufficient to determine the second and the third frequencies with an error below 2%, giving in some case a fully converged result. For the first and fourth frequency, corresponding to shear dominated modes, the number of computations increase by one unit for the coarsest mesh, whereas for the other meshes, one or two computations are still sufficient to assure an almost converged

Figure 3.6: Convergence of the first four lower frequencies versus degrees of freedom for the shell and solid models of the sandwich panel.
Figure 3.7: Convergence of the relative error for the first four lower frequencies versus degrees of freedom for the shell and solid models of the sandwich panel.

result.

Figure 3.7 shows that the behavior of the shell models is much better when it is considered from an engineering point of view and the errors are comparable with those obtained using solid p-elements. Nevertheless, a closer look at the pictures reveals that:

- for the first and the fourth modes, to achieve the same accuracy as the solid p-elements, the number of computations required by the shell models is higher than those required by the solid models, specifically 5 or 6 computations for the shell against 1 or 2 computations for the last two finer solid models. Whereas using p-elements successive computations can be performed in an automatic manner, each successive computation made with shells requires the generation of a new mesh and the set-up of a new analysis, increasing the overall computation time by the amount of time needed to set-up a new case;

- for the second and third modes the error obtained with shell models is comparable with the error obtained with solid models, even using less degrees of freedom. Though it must be observed that the convergence curves
for the shell models are not monotonic. Therefore solutions belonging to the non-monotonic part of these curves cannot be considered for comparison with the solid modes. Under this premise, the number of degrees of freedom corresponding to a solution guaranteed to converge towards the right asymptotic value and therefore acceptable becomes comparable for the solid and shell models. On the other hand, it must be remarked that the convergence curves for the solid models are always monotonic and therefore converged values can be defined using some extrapolation technique as the Richardson’s \cite{45}. Finally, it must be remarked again that for each computation a new shell mesh must be generated, whereas the solid model mesh remains unchanged.

As an appendix it should be pointed out that all the analyses have been carried using a consistent mass matrix model. Linear finite elements are available in the lumped mass matrix formulation, and in the industrial practice they are used in this form in order to take into account the non-structural masses. Nastran solid p-elements allow only the formulation with consistent mass matrix. For the sake of clarity in the Fig. 3.8a–3.8b the same analysis previously done are reported including plate model created using a lumped mass matrix. As can be expected the lumped mass matrix model is much slower than the consistent one, enforcing the conclusion drawn above.

### 3.4 Conclusions and recommendations

The results presented show how the solid p-elements can be used to analyse thin walled structural elements, made of composite materials, without using a detailed model for the laminate. Equivalent models can be defined, based on the use of a single-layered laminate made of a homogeneous and orthotropic material, matching the extensional and shear stiffness of the structural element to be modeled. The equivalent models give the same results as the full multi-layered
shell models. Equivalent models made of solid p-elements give more accurate results using less degrees of freedom than models made of traditional linear shell elements.
Chapter 4

Equivalent Models for Stiffened Panels

4.1 Introduction

Aircraft wings are normally thin walled structures for which the skin needs to be stiffened to prevent buckling at too low stress levels. This stiffening is done with ribs and stringers. When performing a modal analysis on such a wing structure a proper modelling of the stiffeners is required. This means that the contribution of the stringers has to be taken into account. For the purpose of performing a modal analysis, a detailed level of modeling is not necessary, as the quantities that must be captured are not the stresses, but global quantities like frequencies and mode shapes. Performing a detailed modeling of the stringers can lead to unnecessary sophisticated models with a lot more degrees of freedom than necessary. Therefore a procedure for the creation of (dynamically) equivalent models is defined.

The equivalencing procedure follows the general lines of the work from Collier [46], [47]. In the derivation of an equivalent model this author treats a stiffened panel as a laminate, made of different laminae. Each lamina is considered as made of an homogeneous material. In this approach the stiffeners are split into different straight pieces, each of them considered as a lamina of the global laminate, see Fig. 4.1. Following the laminate theory, an ABD matrix [42] can be determined for the whole stiffened panel and then this matrix can be used to define a finite element equivalent model made of only one (laminated) panel. Its elastic behavior is defined assigning the ABD matrix instead of defining elastic properties to the elements of the model. In the Collier approach the effort of the equivalent modelling consists in the determination of the ABD matrix of the stiffened panel which in turn defines the equivalent model.

In the present approach a stiffened panel is still considered as a laminate, but due to the use of brick finite elements, the ABD matrix formulation cannot be used directly because an assumption is needed for the displacement field along the thickness of each layer, like in the shell theory model. Instead a stiffness
Equivalent Models for Stiffened Panels

matrix, at material level, must be defined for each layer which the (equivalent) panel is made of.

The equivalencing process for the bricks is made of two steps:

- the first consists in defining and evaluating suitable “global” properties which, matched by the equivalent model, assure that frequencies and modal shapes of the original model are correctly captured. These global properties are the extensional, bending and torsional stiffnesses, defined according to the engineering beam theory and Bredt torsion theory [48]. Indeed from a stiffened panel, a basic repeating element can be extracted, see Fig. 4.1, for which the engineering bending theory can be applied, whereas for the torsional stiffness evaluation, the Bredt’s theory can be applied to the wing-box section considered. Moreover, for a dynamic analysis is important that the equivalent model matches the total mass of the stiffened panel;

- the second step consists in translating the computed mass and stiffnesses into material and geometric (thicknesses) properties of a laminate made of a suitable number of layers and different materials, such as the frequencies and the modal shapes of the original model are matched.

The two steps will be explained in the following sections and they will be applied to the determination of the equivalent model for a stiffened panel of an aircraft wing-box.

4.2 Global stiffness determination for stiffened panels

The procedure for the determination of the “global” stiffnesses is applied to a stiffened, zero-curvature panel with a sequence of T-shaped stringers as shown in Fig. 4.2. From the panel a cell can be extracted and on that element the equivalencing procedure can be applied. The same method can be applied to curved panels which can be considered as made of stepwise straight pieces. In the
4.2. Global stiffness determination for stiffened panels

4.2.1 Orthotropic materials

In order to derive a general formulation for the evaluation of the global stiffnesses, we make the hypothesis that the skin and the stringers are made of a homogeneous and orthotropic material. In general any stringer cross-section, see Fig. 4.3, can be seen as built from straight pieces or divided into piecewise straight parts. Every straight part of the cross section is a laminate made of a certain number of laminae. Once assigned the properties, the orientation of each lamina and the lamina’s stacking sequence, the properties of each laminate (i.e. straight parts) can be described by the A, B and D matrices derived from the laminate theory. For the basic repeating element shown in Fig. 4.2, the A, B and D matrices can be defined for the skin (dimensions \(b, t_1\)), and each of the straight elements of the stringer cross-section (dimensions \(b_2, t_2\) and \(b_3, t_3\)).

The total extensional, bending and torsional stiffnesses in the plane containing the cross-section of the basic repeating element can be determined summing up the contribution of the different pieces considered, taking the appropriate elements of the A, B and D matrices. The total extensional stiffness, often denominated \((EA)\) but in a more general context defined now as \((A_{11})_{total}\), is

![Different types of stringer shapes.](image)
Equivalent Models for Stiffened Panels

given by:

\[
(A_{11})_{\text{total}} = b_1 A_{11}^{sk} + b_2 A_{11}^{str2} + b_3 \frac{1}{(A_{str3}^{-1})_{11}},
\]

where \(A_{11}^{sk}\) is the 11 term of the \(A\) matrix for the layered skin, \(A_{11}^{str2}\) is the 11 term of the \(A\) matrix for the part of the stringer attached to the skin (dimensions \(b_2, t_2\)) and \(A_{str3}\) is the \(A\) matrix for the vertical part of the stringer (dimensions \(b_3, t_3\)), see Fig. 4.2b.

The meaning of the last term between parentheses can be explained remembering that the components of the extensional stiffness matrix of the laminate \(A_{ij}\) define the following relation [42] expressed in the frame of reference of Fig. 4.4 for the part of the stringer attached to the skin (dimensions \(b_2, t_2\)):

\[
\begin{bmatrix}
N_z \\
N_x \\
N_{zx}
\end{bmatrix} = \begin{bmatrix}
A_{11} & A_{12} & A_{16} \\
A_{12} & A_{22} & A_{26} \\
A_{16} & A_{26} & A_{66}
\end{bmatrix} \begin{bmatrix}
\epsilon_0^z \\
\epsilon_0^x \\
\gamma_0^{zx}
\end{bmatrix},
\]

where \(z \equiv 1, x \equiv 2\) and \(N_z, N_x, N_{zx}\) are the forces (per unit width) acting on the laminate and \(\epsilon_0^z, \epsilon_0^x, \gamma_0^{zx}\) are the constant part of the extensional and shear deformation of the laminate, assumed linear according to the laminate theory.

Similarly, for the vertical part of the stringer (dimensions \(b_3, t_3\) of Fig. 4.2b), the following relation holds:

\[
\begin{bmatrix}
N_z \\
N_y \\
N_{zy}
\end{bmatrix} = \begin{bmatrix}
A_{11} & A_{12} & A_{16} \\
A_{12} & A_{22} & A_{26} \\
A_{16} & A_{26} & A_{66}
\end{bmatrix} \begin{bmatrix}
\epsilon_0^z \\
\epsilon_0^y \\
\gamma_0^{zy}
\end{bmatrix},
\]

with the same meaning for \(N_z, N_y, N_{zy}\) and \(\epsilon_0^z, \epsilon_0^y, \gamma_0^{zy}\), and where \(z \equiv 1, y \equiv 2\).

Figure 4.4: Section of the stringer considered for the evaluation of the stringer’s contribution to the extensional, bending and torsional stiffness of the stiffened panel in the plane perpendicular to the \(z\) axis.
In the following, it is assumed that the stringer contributes to the increase of the skin’s structural stiffness only along the stringer axis direction (z-axis of Fig. 4.2), therefore only through the $A_{11}$ term of the $A$ matrix.

The $A_{11}$ term describes the force per unit width that appears when a unit strain in the $z$-direction, $e_{0}^{z}$, is applied in absence of strains $e_{0}^{x}$ and $\gamma_{0}^{xz}$ along the $x$ and $zx$ directions for the horizontal part of the stringer, and in absence of strains $e_{0}^{y}$ and $\gamma_{0}^{yz}$ along the $y$ and $zy$ directions for the vertical part of the stringer.

The part of the stringer attached to the skin (shown in Fig. 4.2b with dimensions $b_{2}, t_{2}$), can satisfy the condition $e_{x}^{0} = \gamma_{xz}^{0} = 0$.

The vertical part of the stringer (shown in Fig. 4.2b with dimensions $b_{3}, t_{3}$) is free to contract and therefore $e_{y}^{0}$ and $\gamma_{yz}^{0}$ cannot be zero. Instead $N_{y}$ and $N_{zy}$ can be assumed zero. Indeed the equivalent model is used to mimic the behaviour of the stiffened panel in the context of a wing-box. The skin is carrying normal forces and is stiffened to prevent it from buckling when loaded in compression. It can be assumed that the stiffened panel is not loaded with out-of-plane shear, therefore the $N_{xy}$ is zero. The $N_{y}$ describes the load per unit width perpendicular to the generator of the stringer. This is an unloaded free edge, so $N_{y}$ is zero by nature, therefore the contribution to the panel axial stiffness can be determined using the relation:

$$
\begin{bmatrix}
\epsilon_{x}^{0} \\
\epsilon_{y}^{0} \\
\gamma_{xz}^{0}
\end{bmatrix} =
\begin{bmatrix}
(A^{-1})_{11} & (A^{-1})_{12} & (A^{-1})_{16} \\
(A^{-1})_{12} & (A^{-1})_{22} & (A^{-1})_{26} \\
(A^{-1})_{16} & (A^{-1})_{26} & (A^{-1})_{66}
\end{bmatrix}
\begin{bmatrix}
N_{x} \\
N_{y} \\
N_{zy}
\end{bmatrix}
$$

(4.4)

The reciprocal of the term $(A^{-1})_{11}$ of the flexibility matrix, used in eq. (4.4) therefore defines better than the $A_{11}$ term itself the contribution of the vertical part of the stringer to the skin’s structural stiffness along the $z$-axis of Fig. 4.2.

The total bending stiffness can be evaluated remembering that the stringer is considered as a mono-dimensional structural element and, it contributes through its extensional stiffness to the total bending stiffness. Defining the total bending stiffness as $(D_{11})_{total}$, it results that:

$$(D_{11})_{total} = b_{1}D_{11}^{sk} + b_{2} \left[ \frac{A^{str2}}{t_{2}} \right]_{11} \int_{y_{2}^{str}}^{y_{2}^{str2}} (y - y_{0})^{2} dy +
\frac{1}{t_{3}} \left[ \frac{A^{str3}}{b_{3}} \right]_{11}^{-1} \int_{y_{2}^{str3}}^{y_{2}^{str32}} (y - y_{0})^{2} dy,$$

(4.5)

where $D_{11}^{sk}$ is the 11 term of the $D$ matrix for the layered skin, $A^{str2}$ is the $A$ matrix for the part of the stringer attached to the skin (dimensions $b_{2}, t_{2}$), $A^{str3}$ is the $A$ matrix for the vertical part of the stringer (dimensions $b_{3}, t_{3}$). $y_{0}$ is the position of the neutral axis of the skin-stringer assembly cross-section measured with respect to the skin mid-plane and $y$ is the position of a point of the cross-section measured with respect to the skin mid-plane, see Fig. 4.2b.
Equivalent Models for Stiffened Panels

Geometrical information used to build the $D_{11}$ term of the $D$ matrix for the laminated skin.

Geometrical information used to build the $D_{11}$ term of the $D$ matrix for the laminated stringer.

Figure 4.5: Sketch of a laminated skin and a laminated stringer.

The meaning of the different terms in the definition can be explained with the aid of Fig. 4.5 and remembering that the components of the $D$ matrix enter in the equation:

$$
\begin{bmatrix}
M_z \\
M_x \\
M_{zx}
\end{bmatrix} =
\begin{bmatrix}
D_{11} & D_{12} & D_{16} \\
D_{12} & D_{22} & D_{26} \\
D_{16} & D_{26} & D_{66}
\end{bmatrix}
\begin{bmatrix}
k_z^0 \\
k_x^0 \\
k_{xz}^0
\end{bmatrix},
$$

(4.6)

where $M_z, M_x, M_{zx}$ are the moments per unit width and $k_z^0, k_x^0, k_{xz}^0$ are the curvatures defined in the frame of reference $xyz$. With a reasonable accuracy it can be said that the stringer affects only the $D_{11}$ term of the $D$ matrix, relating the moment $M_z$ and the curvature $k_z$, thus the bending stiffness of the skin-stringer assembly around the $x$-axis.

For the skin, the contribution to the total bending stiffness is directly defined through the $D_{11}$ term of the pertinent $D$ matrix, evaluated with respect to the position of the neutral axis of the assembly skin-stringer, therefore:

$$
D_{11}^{sk} = \frac{1}{3} \sum_{k=1}^{N} (Q_{11})_k [(y_k - y_0)^3 - (y_{k-1} - y_0)^3],
$$

(4.7)

where $(Q_{11})_k$ is the 11 term of the material’s stiffness matrix $Q$ for the layer $k$ of the laminate skin, $y_0$ is the position of the neutral axis of the assembly skin-stringer with respect to the reference frame defined in Fig. 4.5 and $y_k$ is the position of the layer $k$ of the laminated skin with respect to that reference frame. The neutral axis position $y_0$ is known and computed from the cell geometry under consideration, defined as the set of points where the deformation perpendicular to the section is zero when the section is loaded with a bending moment.

The stringer contribution to the bending stiffness of the skin-stringer assembly is evaluated using an expression similar to Eq. (4.7), assuming that the stringer affects the only bending stiffness around the $x$-axis of Fig. 4.1. Following the approach introduced in [41], each laminated straight part of the stringer is considered as a one-layered homogeneous material; for the horizontal part of the stringer, of dimensions $(b_2, t_2)$, the axial stiffness is defined as $\overline{Q}_{11} = [A^{str2}/t_2]_{11}$ whereas for the vertical part of dimensions $(b_3, t_3)$, the axial stiffness is defined...
4.3 Equations for the equivalent modelling

Using the same notations introduced in the preceding section, the equations for the equivalencing of the different stiffnesses and the equivalencing of the mass

as $Q_{11} = [(A_*^{str3}/b_3^{-1})]_{11}$. The total bending stiffness evaluation results in the expression of Eq. 4.3.

This approach for the evaluation of the contribution of the skin-stringer assembly is particularly useful when the stringer has a complex curved geometry and therefore it is split into several straight pieces. Alternatively, for a simpler geometry like the one considered in the Fig. 4.5, a direct derivation of the contribution to the bending stiffness can be evaluated exactly, without using the $A$ matrix for each of the pieces. In this case the expression of the bending stiffness contribution becomes:

$$(D_{11})_{total} = b_1 D_{11}^{sk} + b_2 D_{11}^{str2} + D_{11}^{str3},$$

(4.8)

where $D_{11}^{str2}$ is the 11 term of the $D$ matrix for the part of the stringer attached to the skin, dimensions $(b_2, t_2)$, $D_{11}^{str3}$ is the 11 term of the $D$ matrix for the vertical part of the stringer, dimensions $(b_3, t_3)$, see Fig. 4.2b. Referring to Fig. 4.2b, the contribution of the part of the stringer connected to the skin is in this case:

$$D_{11}^{str2} = \frac{1}{3} \sum_{k=1}^{N} (Q_{11}^{str2})_k [(y_k - y_0)^3 - (y_{k-1} - y_0)^3],$$

(4.9)

where $(Q_{11}^{str2})_k$ is the 11 term of the material’s stiffness matrix $Q$ for the layer $k$ of the part of the stringer attached to the skin and the sum is extended over the $N$ layers of the horizontal part of the stringer, shown in Fig. 4.5b. In particular, $y_1 = Y_1^{str2}$ and $y_N = Y_N^{str2}$.

Similarly, for the vertical part of the stringer, Fig. 4.5b, it results that:

$$D_{11}^{str3} = \frac{1}{3} \left[(Y_2^{str3} - y_0)^3 - (Y_1^{str2} - y_0)^3\right] \sum_{k=1}^{N} (Q_{11}^{str2})_k (x_k - x_{k-1}),$$

(4.10)

The different geometric quantities introduced in the formula are inserted in the Fig. 4.5b.

The total mass, defined as $(\rho A)_{total}$, is given by:

$$(\rho A)_{total} = \rho^{str} A^{str} + \rho^{sk} A^{sk},$$

(4.11)

where $\rho^{str}$ and $\rho^{sk}$ are the mass densities of the stringer and the skin respectively, $A^{str}$ and $A^{sk}$ are the cross sectional areas of the stringer and the skin respectively.

The evaluation of the total torsional stiffness, defined as $(GJ)_{total}$ is explained in detail in section 4.3.4.

4.3 Equations for the equivalent modelling

Using the same notations introduced in the preceding section, the equations for the equivalencing of the different stiffnesses and the equivalencing of the mass
can be summarized as:

\[(D_{11})_{\text{total}} = (D_{11})_{\text{equiv}},\]  
\[(A_{11})_{\text{total}} = (A_{11})_{\text{equiv}},\]  
\[(GJ)_{\text{total}} = (GJ)_{\text{equiv}},\]  
\[(\rho A)_{\text{total}} = (\rho A)_{\text{equiv}},\]

where \((D_{11})_{\text{equiv}}, (A_{11})_{\text{equiv}}, (GJ)_{\text{equiv}}, (\rho A)_{\text{equiv}}\) are the bending stiffness, the axial stiffness, the torsional stiffness and the total mass, respectively, for the equivalent models that will be defined in the following sections.

From this point on, we assume that the equivalent models will be based on orthotropic materials, with the principal frame of reference coincident with the one shown in Fig. 4.5. The only elastic properties that will be affected by the equivalencing process are those defined in a section orthogonal to the stringer direction, thus the properties defined in the plane of the stringer’s section. The above equations can be decoupled at first instance. Indeed, firstly Eqs. (4.12) and (4.13) (bending and axial stiffness equivalence) can be solved. In fact they are coupled by the material’s axial stiffness (Young’s modulus in the isotropic case). As can be seen later on in this chapter, the solution of these two equations provides the geometry of the smeared stringer as solution. Secondly, the torsional equation (4.14) can be solved to determine the shear shear modulus in the plane of the section. Finally, the mass equation (4.15) can be solved to determine the equivalent density.

In the following, different approaches are presented to build an equivalent model and to solve the equations derived in the previous section.

### 4.3.1 One layer smeared stiffener approach

In this first approach the equivalent model is represented by a two-layers plate, Fig. 4.6, where the first layer, of thickness \(t_{1e} = t_{1}\), is the skin, and the second layer, with thickness \(t_{3e}\), is the “smeared” stringer.
The unknowns of the problem are the thickness $t_{3e}$, the Young’s modulus of the smeared stringer in the stringer direction $E_{3e}$, the density of the smeared stringer $\rho_{3e}$, the shear modulus of the smeared stringer $G_{3e}$ and the axial stiffness of the skin $E_{1e} = E_{1e}/(1 - \nu_{zz}^s \nu_{zz}^r)$, $E_{1e}$ being the equivalent Young modulus of the skin and $\nu_{zz}^s, \nu_{zz}^r$ the Poisson’s ratios relating the axial strains along the directions $x$ and $z$ of Fig. 4.6.

In total there are 5 unknowns, thus one more than the number of equations 4.12-4.15 written above. One extra equation is needed, which is related to the requirement that the equivalent model keeps the same position of the neutral axis of the original model, see Fig. 4.2c. The additional equation reads:

$$ (y_n)_{total} = (y_n)_{equiv} \ , $$  \hspace{1cm} (4.16)

where $(y_n)_{total} = y_0$ is the position of the neutral axis of the assembly skin-stringer, shown in Fig. 4.2c and $(y_n)_{equiv}$ is the position of the neutral axis of the equivalent model, see Fig. 4.6b.

It must be pointed out that the elastic properties that have been referred to, are the properties in the section considered. With this regard, the equivalent model for the assembly skin-stringer is considered made of an orthotropic material. As remarked above, the first equations to be solved are those involving the axial and the bending stiffnesses:

$$ (A_{11})_{total} = (A_{11})_{1equiv} + (A_{11})_{3equiv} \ , $$  \hspace{1cm} (4.17)

$$ (D_{11})_{total} = (D_{11})_{1equiv} + (D_{11})_{3equiv} \ . $$  \hspace{1cm} (4.18)

Starting point are Eqs. 4.17 and 4.18. From Eq. 4.17 it follows that:

$$ (A_{11})_{total} = \frac{(A_{11})_{total}}{t_{1e}} b t_{1e} + \frac{E_{3e}}{b t_{3e}} b t_{3e} \ , $$  \hspace{1cm} (4.19)

with $t_{1e} = t_1$ ($t_1$ is the thickness of the skin), $E_{3e} = E_{3e}/(1 - \nu_{zz}^s \nu_{zz}^r)$ and $\nu_{zz}^s, \nu_{zz}^r$ are the Poisson’s ratios relating the axial strains along the directions $x$ and $z$ of Fig. 4.6b. It follows that:

$$ E_{1e} = (A_{11})_{total} b t_{1e} \ , $$  \hspace{1cm} (4.20)

The position of the neutral axis is defined by:

$$ y_n = \frac{y_{3e} (A_{11})_{3equiv}}{(A_{11})_{total}} \ , $$  \hspace{1cm} (4.21)

with

$$ y_{3e} = \frac{t_1}{2} + \frac{t_{3e}}{2} \ , $$  \hspace{1cm} (4.22)

and

$$ (A_{11})_{3equiv} = E_{3e} b t_{3e} \ . $$  \hspace{1cm} (4.23)

This yields:

$$ y_n = \frac{(A_{11})_{total}}{y_{3e} b t_{3e} E_{3e}} = \frac{t_1}{2} + \frac{t_{3e}}{2} \ , $$  \hspace{1cm} (4.24)
which results in:

\[ b t_{3e} E_{3e} = \frac{2 y_n (A_{11})_{total}}{t_1 + t_{3e}} = \frac{K_0}{t_1 + t_{3e}}, \]  

(4.25)

with \( K_0 = constant = 2 y_n (A_{11})_{total} \), \( y_n \) and \( (A_{11})_{total} \) being known from the full model. From the bending equation, with the inertia moments evaluated around the neutral axis we have:

\[(D_{11})_{total} = (D_{11})_{1e} + (D_{11})_{3e} = E_{1e} I_1 +
E_{3e} \left( \frac{b t_{3e}^3}{12} + (y_{1e} - y_n)^2 A_{3e} \right), \]

(4.26)

\( I_1 \) being known from the full model. Replacing \( E_{1e} \) in the above equation with expression 4.20 and replacing \( y_{3e} \) with expression 4.22:

\[(D_{11})_{total} = (A_{11})_{total} - E_{3e} b t_{3e} I_1 +
E_{3e} \left( \frac{b t_{3e}^3}{12} + (y_{1e} - y_n)^2 A_{3e} \right). \]

(4.27)

Replacing the product \( E_{3e} b t_{3e} \) with expression 4.25, the final equation to be solved appears as:

\[(D_{11})_{total} = \frac{(A_{11})_{total}}{b t_1} I_1 - \frac{K_0}{b t_1 (t_1 + t_{3e})} I_1 + \frac{K_0}{t_1 + t_{3e}} \frac{t_{3e}^2}{12} +
\frac{K_0}{t_1 + t_{3e}} \left( \frac{t_1 + t_{3e}}{2} - y_n \right)^2. \]

(4.28)

Eq. 4.28 is a non-linear equation in the variable \( t_{3e} \), and can be solved using an iterative procedure. Once solved for \( t_{3e} \), the value of \( E_{3e} \) can be determined from Eq. 4.25 and then the value of \( E_{1e} \) can be determined from Eq. 4.20.

### 4.3.2 One layer smeared stringer simplified approach

The equivalencing procedure presented in section 4.3.1 requires that the neutral axis of the basic repetitive element doesn’t change position. Equivalent models based on a one-layer approach can be derived just assuming that the neutral axis can be free. This is motivated by the fact that reducing by one the number of variables could avoid the modification of the material properties of the skin, and the equivalent model could be more easily implemented at finite element level. In addition, some tests have shown that keeping free the neutral axis position would keep the thickness of the smeared stringer small, within the boundaries of the thin-walled structural elements behavior. Preventing the use of “thick” solid finite-elements could avoid the modeling of unrealistic shear-bending couplings.

The equation to be solved remains non-linear but removing the constraint has an effect on the smeared stringer model, which has a thickness smaller than the thickness of the model that keeps the neutral axis. In this case the number
of unknowns are reduced by one and they are limited to the thickness of the equivalent layer \( t_{3e} \), the axial stiffness of the equivalent layer \( E_{3e} \), the density of the smeared stringer \( \rho_{3e} \) and the shear modulus of the smeared stringer \( G_{3e} \).

Starting from the axial and bending stiffness equivalence, Eqs. 4.17-4.18, the axial stiffness equation can be written as:

\[
(A_{11})_{total} = E_{1} b t_{1} + E_{3e} b t_{3e}.
\]  

(4.29)

From this it follows that:

\[
b E_{3e} t_{3e} = (A_{11})_{total} - E_{1} b t_{1} = constant. \]

(4.30)

The position of the neutral axis of the equivalent layer \( y_{3e} \) with respect to the midplane of the skin is defined by:

\[
y_{3e} = \frac{t_{1}}{2} + \frac{t_{3e}}{2}. \]

(4.31)

The position of the neutral axis of the skin-equivalent layer assembly with respect to the midplane of the skin is defined by:

\[
y_{n} = \frac{y_{3e} (A_{11})_{3equiv}}{(A_{11})_{total}}, \]

(4.32)

\( (A_{11})_{total} \) being known and resulting \( (A_{11})_{3equiv} = b E_{3e} t_{3e} = constant \), from Eq. 4.30 the neutral axis position of the equivalent panel and stringer assembly can be written as:

\[
y_{n} = K_{1} y_{3e} = K_{1} \frac{(t_{1} + t_{3e})}{2}, \]

(4.33)

with

\[
K_{1} = \frac{(A_{11})_{3equiv}}{(A_{11})_{total}}. \]

(4.34)

The bending stiffness equivalence, Eq. 4.18 can be written as:

\[
(D_{11})_{total} = E_{1} \left( \frac{b t_{1}^{3}}{12} + A_{1}(y_{n} - y_{1})^{2} \right) + E_{3e} \left[ \frac{b t_{3e}^{3}}{12} + A_{3e}(y_{3e} - y_{n})^{2} \right] \]

\[
= E_{1} \frac{b t_{1}^{3}}{12} + E_{1} A_{1} y_{n}^{2} + E_{3e} \frac{b t_{3e}^{3}}{12} + E_{3e} A_{3e}(y_{3e} - y_{n})^{2} \]

\[
= E_{1} \frac{b t_{1}^{3}}{12} + E_{1} b t_{1} y_{n}^{2} + E_{3e} b t_{3e} \frac{t_{3e}^{2}}{12} + E_{3e} b t_{3e}(y_{3e} - y_{n})^{2}. \]

(4.35)

Substituting Eq. 4.33 into Eq. 4.35 and defining \( K_{2} = constant = E_{3e} b t_{3e} \) it
follows that:

\[(D_{11})_{\text{total}} = \frac{E_1 b t_1}{12} t_3 e + \frac{E_1 b t_1}{12} K_1 \left(\frac{t_1 + t_3 e}{2}\right)^2 + K_2 \frac{t_3 e^2}{12} + \frac{K_2 (y_3 e - K_1 y_3 e)^2}{2} + \frac{K_2 t_3 e^2}{12} + K_2 \frac{(t_1 + t_3 e)^2}{4} (1 - K_1)^2.\]

Equation 4.36 is a non-linear equation of the (only) variable \(t_3 e\). Solving for \(t_3 e\), from Equation 4.36 the value of \(E_3 e\) can be determined.

### 4.3.3 Mass Equivalence

The total mass of the basic repeating element, Fig. 4.2b must equal the mass of the equivalent model, Fig. 4.6. Thus it comes out that:

\[\rho_{\text{str}} A_{\text{str}} = \rho_{\text{eq}} (t_3 e) b,\]

where \(\rho_{\text{str}}\) is the density of the stringer and \(A_{\text{str}} = (b_2 t_2)(b_3 t_3)\) is the area of the front section of the stringer. The equivalent density \(\rho_{\text{eq}}\) is given from:

\[\rho_{\text{eq}} = \rho_{\text{str}} \frac{A_{\text{str}}}{(t_3 e) b}.\]

### 4.3.4 Shear stiffness equivalence

Differently from the equivalencing procedure followed to determine the thickness and the axial stiffness of the smeared stringer, which involved only the properties of the basic repeating element, the determination of the shear modulus of the smeared stringer needs the estimation of the torsional stiffness of the complete wing-box to be analysed.

Starting point is the Lodet’s torsion theory for closed sections, exposed in the De St Venant problem for a solid 48. In our case the closed section solid to be considered is the wing-box section and the area enclosed in it. For simplicity, the equivalencing procedure is developed for a wing-box having straight parts as shown in Fig. 4.7, in order to eliminate the errors connected to the geometry approximation of curved structures.

Particular attention must be posed in the calculation of the torsional stiffness of the full model in presence of the stringers. Indeed the presence of the stringers gives a certain contribution to the torsional stiffness of the wing-box. This contribution must be evaluated in some way. According to Lodet’s theory,
a torsional moment is resisted by a closed section through a shear flow running along the thickness of the section. This shear flow is constant along the thickness of each point of the section.

The (mean) shear stress at each point of the closed section is expressed as:

\[ \tau = \frac{M}{2At} \]  

(4.39)

where \( \tau \) is the mean shear stress, \( M \) is the applied moment, \( A \) is the area enclosed by the section and \( t \) is the local thickness of the section. It is reasonable to consider that the only contribution is given from the part of the stringer that is in contact with the skin (the flange). This is particularly true for a glued flange, whereas for a riveted flange the contribution of the stringer can be neglected. For a glued stringer, for the evaluation of the torsional stiffness, the basic repeating element made from the skin and the stringer does not appear as in Fig. 4.2b but as in Fig. 4.8a. Moreover, the part of the stringer considered can be smeared along the stringer pitch such as that the area \( bt_2 \) remains constant. In this way the computation of the area \( A \) is easier. The result of the smearing is shown in Fig. 4.8a where the flange of the stringer has now a new thickness \( t_{2sm} \) defined by:

\[ t_{2sm} = t_2 \frac{b_2}{b} \]  

(4.40)

The estimation of the torsional stiffness of the full model follows the steps of the Bredt’s theory. Starting point is the observation that the flow over a section,
Equivalent Models for Stiffened Panels

(a) Effective shape of the wing-box considered for the torsional stiffness determination.

(b) Shear flow over the equivalent effective section of figure 4.8b and definition of the local integration coordinates \(d\bar{x}_1\), \(d\bar{x}_2\).

Figure 4.9: Effective shape of the wing-box and shear flow definition for the determination of the torsional stiffness.

represented by the two layers in Figs. 4.8b and 4.9b is constant, thus:

\[
\int_1 \tau dt = \text{constant}, \quad \text{with} \quad t = t_1 + t_{2sm}.
\]

The shear stress changes along the thickness of the section considered and of course in the skin and in the stringer the shear stresses are in general different. Following the Bredt’s theory, without loss of generality we can refer to a mean stress \(\bar{\tau}\) along the section considered, and Eq. 4.41 can be written as:

\[
\bar{\tau} t = \bar{\tau} (t_1 + t_{2sm}) = \text{constant}.
\]

If the skin and the stringers are made of orthotropic materials, then the relation between the torsional moment \(M\), the rotation per unit length of the section \(\theta\), the mean shear stress \(\bar{\tau}\) and the shear modulus of two materials \(G_1\) and \(G_2\) becomes:

\[
M \theta = \int_{\partial A} \frac{\bar{\tau}^2}{G} t d\bar{s} = \int_{\partial A_1} \frac{\bar{\tau}^2}{G_1} t d\bar{s} + \int_{\partial A_2} \frac{\bar{\tau}^2}{G_2} t d\bar{s},
\]

where the integrals are determined over the total area where the shear stresses run, denoted in Fig. 4.9b by the oblique lines (area \(A_1\), the wing-box area) and the vertical lines (area \(A_2\), the smeared effective stringer area), \(\bar{\tau}\) being the curvilinear coordinate running along the wing-box section. The integrals over the areas can be transformed into line integrals as we are considering a mean shear stress over the thicknesses:

\[
M \theta = \int_{\bar{x}} \frac{\bar{\tau}^2}{G_1} t d\bar{x}_1 + \int_{\bar{x}} \frac{\bar{\tau}^2}{G_2} t_{2sm} d\bar{x}_2.
\]

As the geometry of each component (skin, stringers, spars) is straight the running
4.3. Equations for the equivalent modelling

coordinates $\mathcal{T}_1$ and $\mathcal{T}_2$ are the same, thus:

$$ M \theta = \oint_{\mathcal{T}} \left( \frac{\tau^2}{G_1} t_1 + \frac{\tau^2}{G_2} t_{2sm} \right) \, d\mathcal{T}, $$

$$ = \oint_{\mathcal{T}} \left( \frac{\tau^2}{G_1} t_1^2 l_1 + \frac{\tau^2}{G_2} t_{2sm}^2 l_1 \right) \, d\mathcal{T}, $$

$$ = \oint_{\mathcal{T}} \left( \frac{\tau^2}{G_1} t_1^2 l_1 + \frac{\tau^2}{G_2} t_{2sm}^2 l_1 \right) \, d\mathcal{T}. \quad (4.45) $$

From Eq. (4.42) it follows that:

$$ M \theta = (\tau t)^2 \oint_{\mathcal{T}} \left( \frac{t_1}{G_1} t_1^2 + \frac{t_{2sm}}{G_2} t_1^2 \right) \, d\mathcal{T}. \quad (4.46) $$

Combining with Eq. (4.39) it comes out that:

$$ M \theta = \frac{M^2}{4 A^2} \oint_{\mathcal{T}} \left( \frac{t_1}{G_1} t_1^2 + \frac{t_{2sm}}{G_2} t_1^2 \right) \, d\mathcal{T}, \quad (4.47) $$

thus:

$$ \theta = \frac{M}{\hat{J}_t}, \quad \text{with} \quad \hat{J}_t = \frac{1}{4 A^2} \oint_{\mathcal{T}} \left( \frac{t_1}{G_1} t_1^2 + \frac{t_{2sm}}{G_2} t_1^2 \right) \, d\mathcal{T}, \quad (4.48) $$

where $\hat{J}_t$ is the expression of the torsional stiffness for the wing-box made of different materials for the skin and the stringers.

For the particular configuration of the wing-box considered, Fig. 4.9a, the torsional stiffness can be expressed as:

$$ \frac{1}{\hat{J}_t} = \frac{1}{4 A^2} \left[ 2 \left( \frac{t_{skin}}{G_{skin}} t_1 + \frac{t_{2sm}}{G_2} t_1 \right) L_1 + 2 \left( \frac{t_{spar}}{G_{spar}} t_1 \right) (L_s - t_{2sm}) \right], \quad (4.50) $$

where $L_1$ and $L_s$ are the chord length and spar height, as shown in Fig. 4.9a and $t_{spar}, G_{spar}$ are the thickness and the shear modulus of the spar. Moreover $t_c = t_{skin} + t_{2sm}, t_s = t_{spar}$ and $A = L_c (L_s - t_{2sm}).$

Once the torsional stiffness of the full wing-box model has been evaluated using Eq. (4.50) it must be taken care that the equivalent model exhibits the same torsional stiffness. This can be done in several ways, modifying the shear modulus of the skin and/or the equivalent stringers. In the present work it has been decided to alter the pertinent shear modulus of the skin in order to match the torsional stiffness of the full wing-box model, and assign a value to the shear modulus of the equivalent stringer such as it does not have any effect on the torsional stiffness of the equivalent wing-box. This choice has been motivated by the fact that as it will be seen from the results, the equivalent smeared stringer has a thickness that is one order of magnitude higher than the thickness of
the skin. This bigger thickness can produce couplings that don’t appear in a thin-walled structural element. In order to avoid this effect that would lead to model behavior that in the full model are not present, it has been decided to characterize the equivalent stringer for the behavior that must be modeled and avoid giving properties that can be assigned to the skin only. According to the strategy shown above the equivalencing procedure is expressed in the following way:

\[
\frac{1}{J_{eq}} = \frac{1}{J_t} = \frac{1}{4A^2} \int_{c_t} \frac{t}{Gt_{tot}} \, d\tau,
\]

\[
= \frac{1}{4A^2} \int_{c_t} \frac{1}{Gt} \, d\tau,
\]

\[
= \frac{1}{4A^2} \left[ \frac{2}{G_{eq} t_{skin}} L_c + \left( \frac{2}{G_{spar} t_{spar}} \right) L_s \right]. \tag{4.51}
\]

The equivalent shear modulus \(G_{eq}\) is finally defined by the following expression:

\[
\frac{1}{G_{eq}} = \frac{t_{skin}}{2L_c} \left[ \frac{4A^2}{J_t} - \left( \frac{2}{G_{spar} t_{spar}} \right) L_s \right] \tag{4.52}
\]

It must be remarked that the procedure exposed above for the shear stiffness equivalence is applicable to any kind of stringer shape except for corrugated panels. In this case a similar procedure can be developed using the Bredt’s theory and taking into account the increase in the area of the resisting section as shown in Stroud \[50–52\].

### 4.4 Wing-box analysis

The equivalencing procedure is now tested on a wing-box model. The wingbox has the rectangular section shown in Fig. 4.7a, which has been chosen in order to avoid all the errors connected with the approximation of a structure with curvature and thus limit the differences between the full and the original model to the equivalencing modeling errors. The wing-box is provided with ribs and stringers attached to the upper and bottom skins.

The wing-box has a chordwise dimension \(L_c = 400 \text{ mm}\), height \(L_s = 100 \text{ mm}\) and a span of \(L_{span} = 6000 \text{ mm}\). The thicknesses of the skin, spars, ribs and stringer components are constant and equal to \(t = 1.3 \text{ mm}\). The wing-box is made of an isotropic material with Young’s modulus \(E = 70 \text{ GPa}\), Poisson’s ratio \(\nu = 0.33\), and density equal to \(\rho = 2700 \text{ Kg/m}^3\).

The wing-box is provided with seven, T-shaped stringers attached to the upper and bottom skins. The stringers dimensions, defined in Fig. 4.2b are: \(b_2 = 32 \text{ mm}, t_2 = 1.3 \text{ mm}, b_3 = 1.3 \text{ mm}, t_3 = 14.7 \text{ mm}\).

The wing-box cross section dimensions were taken from a wing-box available at the Structures and Materials Laboratory of the Aerospace Engineering faculty, whereas the sizes of the stringers were taken from a manufacturer data sheet available in the book from \[53\].
A first wing-box configuration has been generated using 14 ribs. Three different finite element models, made of brick p-elements, have been considered: the first is a full model, where the stringers have been modeled in detail, the second is based on the reduced model for the stringers, one layer smeared stringer model, of the section 4.3.1 and the third is based on the reduced model for the stringers, one layer smeared stringer simplified model, of the section 4.3.2.

A modal analysis has been performed for each of the three models. The analysis has been carried in an adaptive way, assigning the starting and the ending polynomial orders for the shape functions on each of the three dimensions of a solid finite element. In particular the starting orders are equal to \((p_1, p_2, p_3) = (3, 3, 2)\) where the second order is assigned to the polynomial along the thickness, whereas the ending polynomial orders are \((p_1, p_2, p_3) = (7, 7, 2)\).

Figure 4.10 shows the convergence history for the frequencies of the first four modes computed in the modal analysis. The first three modes computed are the first, second and third bending mode, whereas the last one computed is the first torsional mode. Each picture shows the convergence history for the full model, the equivalent model based on the single layer approach, and the equivalent model based on the simplified single layer approach. The corresponding curves are named respectively Full-Model, Equiv-Neut, Equiv. Besides, two additional

---

Figure 4.10: Convergence analysis of the first four modes for the 14 ribs wing-box.
Equivalent Models for Stiffened Panels

curves are represented, corresponding to two additional equivalent models non
mentioned in the preceding sections. These models are based on those intro-
duced in the sections 4.3.1 and 4.3.2 with the difference that in the evaluation
of the global properties of the model the part of the stringer attached to the skin
is considered as purely monodimensional, and therefore in the Eqs. 4.1 4.3 the
terms
\[ b_2 A_{11}^{str^2} \text{ and } b_2 \left[ A_{11}^{str^2} \right] \int_{y_1^{str^2}}^{y_2^{str^2}} (y - y_0)^2 dy \]
are replaced respectively with
\[ b_2 \left[ \frac{1}{(A_{11}^{str^2})^{-1}} \right] \text{ and } b_2 \left[ \left( \frac{A_{11}^{str^2}}{t_2} \right)^{-1} \right] \int_{y_1^{str^2}}^{y_2^{str^2}} (y - y_0)^2 dy \]
and the equivalent stringer is considered as a purely mono-dimensional struc-
tural element and no coupling with the transverse direction is present. The
Poisson’s effect is therefore left out and the elastic constants representing the
material properties of the stringer are defined only by the Young’s modulus, the
Shear modulus and the density. The additional curves are named Equiv-Neut-1d,
Equiv-1d in the plots. Figure 4.10 shows that all the converged results of the
different equivalent models differ from the converged results of the full-model
within 2 percent. Moreover, slightly better results are obtained when the equi-
valent model is based on the one that matches the position of the neutral axis,
and for the second and third mode it happens that the 1-d model is the best.

In order to assess the results, the same analysis has been performed on an-
other wing-box, for which the number of ribs has been increased from 14 to 21.
Figure 4.11 show the convergence history for the frequencies of the first four
modes computed in the modal analysis. Again all the results are comprised
within 2 percent in the worst case, being coincident with the results of the full
model for the second and third mode. The results associated to the torsional
mode show more distance from the full model with respect to the 14 ribs case,
but still below 1.5 percent difference. The converged results are achieved by
the equivalent models saving at least one order of magnitude in degrees of free-
dom with respect to the full finite element model. In the development of the
equivalent models, it has been chosen to discard the material stifnesses in the
plane orthogonal to the axial direction of the stringers, assuming that the stiff-
ening effect is purely mono-dimensional. In order to define this property for
the layer representing the equivalent model, the material elastic stifnesses in
these directions must have been to be set to a (numerical) zero value. A zero
value cannot be input otherwise the stifness matrix associated to the finite ele-
ments of the equivalent models can be ill-conditioned, thus a value such that
no influence can be produced on the model is introduced. The material of the
equivalent layer is assumed to be orthotropic and as zero value for the Young
modulus in the transversal direction (\(E_{xx}\)) and for the shear modulus \(G_{xy}\)
in the plane of the section has been taken a value of \(E_{xx} = G_{xy} = 10^6 N/m^2\). The
choice has been done considering as zero value a value at least three orders of magnitude from the smallest elastic constant. In order to verify this choice, a modal analysis has been performed taking as zero reference value the value of $E_{xx} = G_{xz} = 10^7 \text{N/m}^2$, ten times bigger that the previous. Figure 4.12 shows the results of the modal analysis for the full and the equivalent models described in section 4.3.1 adopting the two different zero-reference-value. The converged results of the first three (bending) modes are not affected at all by the variation in these values, whereas the last mode, the torsional, is affected with a variation of the 0.8 percent in the frequency. Thus a variation of one order of magnitude of the zero-reference-value leads to a variation of the torsional frequency by less than one percent which leads to a total difference from the converged value of the full model by less than 2.5 percent, still within the engineering acceptability of the results. Finally, another sensitivity analysis has been performed on the equivalent models. The polynomial order chosen along the thickness of the elements is equal to 2, the minimum admissible to avoid locking effects. In order to verify this fact practically, a modal analysis has been performed, where the polynomial order along the thickness has been set equal to three. Figure 4.13 show the convergence analysis for the two choices for the polynomial orders along the thickness. The curves corresponding to the two equivalent models are just
shifted with respect to each other, providing the same results for each step of the adaptive analysis.

4.5 Conclusions and recommendations

The results presented show that, using brick, p-formulation finite elements for the modal analysis of aeronautical structures, an equivalent model can be efficiently employed for the determination of the modal properties of a wing box, based on a one-layer, smeared representation for the stringers distribution, where the axial, bending, torsional stiffnesses are matched, along with the total mass and the neutral axis position of the basic repeating element. The different approaches presented give results with a difference within 2 percent from the converged results (natural frequencies) of a full model. The existing differences, although all the geometrical approximations have been filtered out due the use of a geometry with no-curvature, can be imputed to three-dimensional nature of the full model. Couplings effects, like the coupling between bending and membrane behavior of the panel, are present whereas in the derivation of the equivalent models they have been neglected. Nevertheless the results are very good.

Figure 4.12: Sensitivity analysis on the choice of the zero reference Shear and Young moduli for the 21 ribs wing-box.
4.5. Conclusions and recommendations

Figure 4.13: Sensitivity analysis on the choice of the polynomial order along the elements thickness for the 21 ribs wing-box.

The efficiency of the equivalencing procedure presented in this paper, further highlights the capabilities of the finite element modelling and analysis method here employed: the use of the p-elements makes possible to perform adaptive analyses, monitoring at each step the degree of accuracy of the solution computed, and even for complex configurations, i.e. a wing-box equipped with structural details such as stiffeners, the analysis set-up is still flexible and the computations controllable. The equivalencing method here introduced guarantees a saving of one order of magnitude in the total number of degrees of freedom of the finite element model which translates into reduced modelling time and possibility to automate the modelling and analysis process.
Chapter 5

Modal Analysis of Wing-Like Structures

5.1 Introduction

In the previous chapters the ability of the solid p-elements in modelling and analyzing thin walled panels having zero, single or double curvature, and made of an isotropic or an orthotropic material has been described. In this chapter the investigation will be extended to a simplified wing structure. In this situation all the advantages and the possible limits of the modelling/analysis approach can be clearly be evidenced. Regarding the advantages it must remarked that:

- the three-dimensional geometry of the element makes it possible to model any kind of geometrical shape;
- the use of the p-formulation finite elements gives the possibility to perform a convergence analysis using only one initial mesh, with an adaptive method. No restrictions are made on the order of polynomials chosen for the shape functions in any local direction within each element, so the most suitable polynomial order can be chosen, for example lower order for edges that run along the thickness, and higher order for the edges that run spanwise.

These advantages translate in the possibility to perform an adaptive analysis using only one type of finite element and therefore the analysis can be automated. For example Fig. 5.1 shows the different options available in Nastran for the three typical elements - beam, shell and solid - used in finite element analysis. As can be seen, when using solid p-formulation finite elements, less options are available and the automation of the finite element analysis becomes easier.

On the other hand, modelling a complex structure like a wing shows some possible drawbacks of this approach. For example the number of degrees of freedom for a p-elements based FE model increases with the square power of the number of elements used, therefore attention must be taken in choosing the appropriate number of elements. Besides, once established a certain meshing...
strategy for a wing, the number of elements of a solid model is higher than the number of elements of a shell model meshed using the same strategy, see Fig. 5.2.

More important, in order to generate a solid mesh, a solid geometrical model is needed. Differently from the CAD models used for the stress analysis of structural details, CAD models commonly used in the structural analysis of components like a wing are made of surface models, i.e. topologically bi-dimensional geometrical objects. The approach presented in this thesis requires the use of solid models, i.e. models described by geometrical entities topologically three-dimensional. The creation of a solid geometric model and the translation of the geometry into a mesh made of brick-type finite elements is a process that requires a high level of control, which cannot be efficiently implemented simply using the parametric modeling features offered by the CAD packages commonly used in the aerospace industry.

5.2 Solid geometry generator

To overcome the problems listed above it has been decided to create a dedicated solid geometry generator, able to support the solid mesh generation, using the ICAD software. ICAD is a Knowledge-Based Engineering system based upon the Lisp programming language. It offers all the features of a CAD modeler, combined with object oriented programming and rules based reasoning, features proper of a KBE tool. A geometric model can be created writing an algorithm like a standard piece of Lisp code, using the geometrical objects and features available from the CAD-engine. The code can be executed in batch mode and therefore it can be used as a tool within a broader environment such as a DEE. Moreover the geometric model can be “upgraded” to be a real product model and not only a geometric model using the KBE features offered by the software.

The basic idea behind the solid geometry generator concept is to create a solid model starting from a surface model, once assigned the thickness and the materials of the different structural parts making an aircraft.
5.2. Solid geometry generator

The solid geometry generator has been conceived as a capability module of the multi-model generator (MMG) being developed within the DAR group of Delft University of Technology. The MMG is able to generate any kind of aircraft configuration by simply assigning values to variables of an input file. The MMG has been built using the parametric modeling approach offered by the ICAD software. Using a parametric modeling approach, different parts called primitives, have been implemented into ICAD. Each primitive represents an elementary unit which an aircraft can be decomposed into, such as engine, fuselage or wing-trunk. The name wing-trunk is used to refer to any kind of lifting surface and its inner structure, such as wing, horizontal/vertical tail, canard. Combining and joining different primitives (by means of connection elements which are themselves also a type of primitive) several different aircraft configurations can be generated. In particular, for the aims of the present work, we are interested in developing a solid geometry model for the wing-trunk primitive. A wing-trunk contains all the elements needed to describe a wing-like component in a fully parametric way. A wing-trunk is conceived to allow definition of some constitutive elements or building blocks which are the upper and lower-skin, the rib and the spar. The wing-trunk in Fig. 5.4 shows all the constitutive elements enlisted. In particular this wing has two spars and three ribs (one at the root, one at the tip and one at the center of the wing) and it will be taken in the following as reference for showing all the operations performed to create a solid mesh. The input data for the solid geometry generator are the surfaces obtained from the wing-trunk generator module of the MMG. These surfaces are collected in a list containing the upper and lower-skins, spars and ribs surfaces. The ribs are decomposed in leading edge rib, trailing edge rib, center-wing rib, whereas a skin is made of a leading edge skin, a center skin and a trailing edge skin.

![Figure 5.2: Difference in the number of elements generated by the same mesh strategies for the shell and solid models.](image)

![Figure 5.3: Examples of primitives and their definition (from [56]).](image)

...
Figure 5.4: Sketch of the basic structural elements making the wing-trunk primitive (top center): lower and upper-skin (mid left and right side), spar (bottom left) and rib (bottom right).

Figure 5.5: Trimetric view of a rib and its basic components: leading edge rib, centre-wing rib, and trailing edge rib.

Figure 5.6: Trimetric view of the lower wing skin and the basic components: leading edge skin, centre skin, and trailing edge skin.

A value for the thickness is associated to each surface and stored in another list. For each surface a solid model is generated by moving each point of the surface along the local normal by a quantity equal to the thickness of the part considered. At the moment only a constant thickness is supported by the solid geometry model generator but ICAD has features able to generate a solid with variable thickness. From a topological point of view each solid (addressed hereafter as solid patch or simply patch) is defined by memorizing the two offset surfaces (addressed hereafter as faces) which has proven to be a method to store information more efficient than generating real solids. Examples of faces generated from the input surfaces are given in Fig. 5.7.
5.2. Solid geometry generator

(a) Solid model for the upper and lower leading edge skin surfaces.
(b) Solid model for the upper and lower center-wing skin surfaces, and front and rear spars.
(c) Solid model for the center-wing rib surface.

Figure 5.7: Solid models generated for different wing-box surfaces. Each solid model is defined by a set of two surfaces, for each input surface of the wing-trunk.

Figure 5.8: Intersection lines (dashed) resulting from the intersection of the upper-skin surface with the ribs and spars faces.

The next step made by the solid generator is the determination of the intersections between the different solids. Each of the two faces of a solid is intersected with the faces of all the other solid components of the wing-trunk. Some examples are given in Fig. 5.5 where the intersection of the upper-skin with the rear spar and one rib is shown, Fig. 5.6 where the leading edge rib is intersected with the upper and lower-skin patches and the front spar, and Fig. 5.7 where the center-wing rib is intersected with the upper and lower center-wing skin, and the faces of the front and rear spars are shown. The intersections found define the transition elements between two different solid components, elements needed to attach the different solid patches during the finite element generation process. At this point the solid model generated is ready for the creation of a solid mesh. It must be remarked that the solid geometry model generator is fully parametric (within the limited amount of geometries that have been tried so-far), therefore, for whatever number and type of structural elements a corresponding solid model can be determined.
5.3 Solid mesh generator

The solid meshing process starts from assigning values to the parameters that control the mesh to be generated. The mesh is controlled by the five parameters shown in Fig. 5.11 and explained in Table 5.1. Four of them ($NLE$, $NCOVER$, $NSPAR$, $NTE$), define the mesh along the edges of the wing profile, and the latter, $NBAY$, controls the mesh spanwise.

Using p-elements only a very coarse mesh is needed. The lower limit for the number of elements that can be used is determined by the polynomial expansion adopted for the description of the curved edges of the finite elements. In the NASTRAN implementation of the p-elements, a third order polynomial expansion is employed for the curved edges. A separate convergence study can

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$NLE$</td>
<td>number of elements between the front-spar and the leading edge</td>
</tr>
<tr>
<td>$NCOVER$</td>
<td>number of elements between two consecutive spars</td>
</tr>
<tr>
<td>$NSPAR$</td>
<td>number of elements along the spar-height</td>
</tr>
<tr>
<td>$NTE$</td>
<td>number of elements between the rear-spar and the trailing edge</td>
</tr>
<tr>
<td>$NBAY$</td>
<td>number of elements between two consecutive ribs</td>
</tr>
</tbody>
</table>
be performed, only for the geometrical representation, in order to select the most appropriate number of elements to be placed on the curved edges.

Once specified a value for each of the five parameters controlling the mesh, each patch is intersected by a number of planes corresponding to the number of elements specified on each edge of the patch considered. An example is given in Fig. 5.12 where one of the upper-skin faces is intersected by planes corresponding to the mesh specification $NLE=1$, $NCOVER=2$, $NTE=1$, $NBAY=2$. The resulting intersection lines (dashed lines in Fig. 5.12) are collected together with those obtained intersecting the different patches with each other. As an example in Fig. 5.12 are reported the collected intersection lines for one of the upper-skin faces.

These intersection lines are split into two sets, corresponding to the lines parallel to the two local directions $u$ and $v$ needed to describe each face, see Fig. 5.13. For each set the lines are ordered in the following way: the lines parallel to the $v$ direction are ordered according to the ascending $u$ coordinate of the intersection point with one of the two edges of the face (edges corresponding

---

**Figure 5.11:** Parameters needed to define a solid mesh. The parameters are defined between two locations indicated by the arrows. The meaning of the parameters is explained in Table 5.1.

**Figure 5.12:** Intersection of the upper-skin with planes corresponding to the mesh parameters $NLE=1$, $NCOVER=2$, $NTE=1$ and $NBAY=2$. 


Intersection lines considered for the meshing process and definition of the local coordinate system $u$ and $v$.

Intersection lines parallel to the $u$ local coordinate

Intersection lines parallel to the $v$ local coordinate

Figure 5.13: Intersection lines split into two sets, corresponding to the lines parallel to the local $u$ and $v$ direction.

to an iso-$v$ line). The same applies for the lines parallel to the local direction $u$. An example is shown in Fig. 5.13.

Then, for each line of the two sets, the intersection with all the lines of the other set is determined, resulting in a list of nodes ordered according to the local frame of reference $[u,v]$. The list of the nodes spans the two faces of a patch starting from the position $(u, v = 0, 0)$ and running along the increasing $u$ direction on the lines $(u, v_i), v_i = 0 \ldots v_{nax}$. An example for the nodes determined on the upper-skin is shown in Fig. 5.13. Examples of the intersection lines and the resulting nodes are reported in Fig. 5.14 for some other patches.

After all the nodes are determined for each patch, the nodes are written in a file, patch by patch.

Another important step in the mesh generation process, is providing the additional geometrical information that is needed to use a third order polynomial description for the edges of the finite-elements, and a bi-cubic polynomial expansion for the faces of the finite-elements.

For this reason on each edge (part of the intersection lines between two successive nodes) two additional points are computed, located at $1/3$ and $2/3$ of the edge (true) length. These additional points together with the nodal points are used to build a third order polynomial representation for the edge. In Fig. 5.14 the nodes (represented with circles) and the additional points for the higher order
5.3. Solid mesh generator

Figure 5.14: Ordered list of the intersection curves parallel to the local v direction of the upperskin surface.

(a) Nodes resulting from the intersection between the lines parallel to the local u direction and the local v direction of the upper-skin.

(b) Nodes on the upper-skin.

Figure 5.15: Intersection lines and nodes resulting from the intersection between the lines parallel to the local u direction and the local v direction of the upper-skin.

gometry (crosses) are reported.

The nodes of the finite elements and the points for the higher order geometry representation are stored in two separate files.

A Fortran procedure has been written in order to generate a Nastran input file starting from these two files. From the file containing the node coordinates, the node coordinates vector, the node topology vector, the finite elements numbering and the topology matrix relating the node numbers to the element numbers are formed for each solid patch. From the file containing the points for the higher order geometry, the points coordinate vector, the edge number vectors and the topology matrix relating the edge numbers to the higher order geometry points vector is formed.
At this point each patch is completely meshed with the higher order geometry associated. But the patches are not connected as the initial procedure has assigned node numbers to each patch separately, thus coincident nodes belonging to different patches have different numbers. Therefore a procedure to perform the “glueing” of the node numbers is implemented, assigning the same number to coincident nodes belonging to different patches.

At the end of the glueing procedure another routine writes the nodes in a Nastran input file which can be directly given to the program. As an example, in Fig. 5.15 the mesh obtained for the test wing-trunk of Fig. 5.11 is reported, using the meshing parameters shown in the same figure.

Figure 5.19 shows a close-up of the intersection between the upperskin, the center-wing rib and the rear spar, highlighting the solid finite elements. In particular, the wing shown in the picture has a span of 3 meters and each solid patch has a constant thickness of 2 millimeters.

Figure 5.20 shows three typical resulting discretized volumes of the structure. These would be normally modeled using three kind of elements, but can also be modeled with the single type solid p-element under study. Based on the edge-length ratios along the three local directions, the solid elements in Fig. 5.20 can be classified as shell-like elements, beam-like elements, point-like elements. In particular the point-like elements can be very useful when non-structural masses need to be modeled. Non-structural masses are usually modeled concentrating them into points. The Nastran implementation of the solid elements does not allow for concentrating masses in the corner nodes. Point-like solid elements can be used to place these masses, by increasing artificially the material density of these elements. In addition, distributed non-structural masses are usually
condensed into the center of gravity of the distributed mass. For example the mass of the fuel for a fuel tank can be concentrated into the tank’s center of gravity, previously determined. In this situation the rotational inertia of the distributed mass can be matched by modeling the center of gravity as a point-like element having a density and dimensions chosen in order to match the total mass and the rotational inertia around the center of gravity of the distributed mass considered. Using beam-like elements the mass can be attached to the relevant point of the structure where the inertial loads must be transferred.
5.4 Applications

The modelling and analysis tool described in the preceding sections has been tested on the modal analysis of wing-trunks. In order to summarize all the operations needed to transform a set of surfaces describing the wing-trunk into a solid finite element model, Fig. 5.21 shows a flow-chart of the different steps described above.

5.4.1 Straight wing

The first test is done on a simple wing-trunk, made of an upper and a lower-skin, three spars and sixteen ribs (including leading edge ribs, wing center ribs and trailing edge ribs). The wing-trunk is straight (zero sweep), with no twist and no taper. The wing-trunk profile is a NACA64A010 with 10% thickness. The chord is \(c = 1\) m and the span is \(s = 10\) m. Following the steps reported in the flow-chart 5.21, a first surface model is generated by the Multi-Model Generator 1.

Next a solid model is created assigning a thickness to the different surfaces. The thickness is chosen constant and it is assumed to be the same for all patches, equal to \(t = 2.01\) mm. A summary of all the geometrical information relative to the straight wing is reported in Table 5.2. Once a solid model is made, a finite element model can be generated. To exploit the high convergence rate of the p-elements it is important that the third order approximation of the geometry is as close as possible to the original model.
Table 5.2: Parameters defining the geometrical configuration of the Straight Wing.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Root Chord</td>
<td>( c_r = 1 \text{ m} )</td>
</tr>
<tr>
<td>Tip Chord</td>
<td>( c_t = 1 \text{ m} )</td>
</tr>
<tr>
<td>Number of ribs</td>
<td>( N_r = 16 )</td>
</tr>
<tr>
<td>Number of spars</td>
<td>( N_s = 3 )</td>
</tr>
<tr>
<td>Span</td>
<td>( s = 10 \text{ m} )</td>
</tr>
<tr>
<td>Skin thickness</td>
<td>( t_{sk} = 2.01 \text{ mm} )</td>
</tr>
<tr>
<td>Sweep angle</td>
<td>( \alpha = 0 \text{ deg} )</td>
</tr>
<tr>
<td>Spar thickness</td>
<td>( t_{sp} = 2.01 \text{ mm} )</td>
</tr>
<tr>
<td>Twist angle</td>
<td>( \beta = 0 \text{ deg} )</td>
</tr>
<tr>
<td>Rib thickness</td>
<td>( t_{rib} = 2.01 \text{ mm} )</td>
</tr>
<tr>
<td>Wing profile</td>
<td>NACA64A010</td>
</tr>
<tr>
<td>Profile Relative thickness</td>
<td>( t% = 10% )</td>
</tr>
</tbody>
</table>

As no study on the capability of the third order reconstruction of the geometry in approximating the geometry of the model has been done, three different meshes have been considered, defining a target element size, initially chosen such as the coarsest mesh possible is obtained, and progressively halving this size. The parameters corresponding to the three different meshes are reported in Table 5.3. Figure 5.22 shows the mesh for the straight wing corresponding to the coarsest mesh, defined as MESH-1 in Table 5.3. In order to perform the most computationally efficient adaptive analysis, it is important to keep the order of the shape functions constant along the shortest edges of the finite element model, i.e. the edges that run along the thickness of the plate-like finite elements, along all the edges of the point-like elements and along the shortest edges of the beam-like elements, as shown in Fig. 5.20. These edges have a size of the order of a millimeter, compared to the size of the span and the chord of the wing-trunk which are of the order of a meter.

The association of specific edges to a constant and lower order shape function can be done in automatic manner. Therefore, a procedure has been implemented, using the software Matlab [60]. This procedure takes as input the parameters that specify the mesh pattern and some geometrical information, like the number of spars and ribs, and outputs some specific Nastran instructions.

Table 5.3: Three different meshes used to perform the analysis of the Straight Wing and maximum size of the three finite-element types appearing in the solid mesh.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>MESH-1</th>
<th>MESH-2</th>
<th>MESH-3</th>
</tr>
</thead>
<tbody>
<tr>
<td>NLE=1</td>
<td>NLE=1</td>
<td>NLE=2</td>
<td></td>
</tr>
<tr>
<td>NCOVER=1</td>
<td>NCOVER=2</td>
<td>NCOVER=4</td>
<td></td>
</tr>
<tr>
<td>NBAY=1</td>
<td>NBAY=1</td>
<td>NBAY=1</td>
<td></td>
</tr>
<tr>
<td>NSPAR=1</td>
<td>NSPAR=1</td>
<td>NSPAR=1</td>
<td></td>
</tr>
<tr>
<td>NTE=1</td>
<td>NTE=1</td>
<td>NTE=3</td>
<td></td>
</tr>
<tr>
<td>Max Plate-Like Element</td>
<td>( 656 \times 348 \times 2.01 \text{ mm} )</td>
<td>( 348 \times 328 \times 2.01 \text{ mm} )</td>
<td>( 174 \times 174 \times 2.01 \text{ mm} )</td>
</tr>
<tr>
<td>Max Beam-Like Element</td>
<td>( 656 \times 2.01 \times 2.01 \text{ mm} )</td>
<td>( 348 \times 2.01 \times 2.01 \text{ mm} )</td>
<td>( 174 \times 2.01 \times 2.01 \text{ mm} )</td>
</tr>
<tr>
<td>Max Point-Like Element</td>
<td>( 2.01 \times 2.01 \times 2.01 \text{ mm} )</td>
<td>( 2.01 \times 2.01 \times 2.01 \text{ mm} )</td>
<td>( 2.01 \times 2.01 \times 2.01 \text{ mm} )</td>
</tr>
<tr>
<td>No. of elements= 607</td>
<td>No. of elements= 862</td>
<td>No. of elements= 1708</td>
<td></td>
</tr>
</tbody>
</table>
Figure 5.22: Finite element model of the Straight Wing corresponding to the coarsest mesh (MESH-1 in Table 5.3). Only the connectivity between the corner nodes is shown, without the points for the higher order geometry approximation.

Figure 5.23: Node numbering of a solid element and definition of the local polynomial orders $p_1$, $p_2$, $p_3$ along the three local coordinate directions (highlighted in boldface) for the association of the edges to specific order shape functions.

that define three different groups of elements. These groups contain respectively the plate-like elements, the point-like elements and the beam-like elements of the FE model. For all the groups, the minimum and maximum order of the shape functions is specified, associating each of the three local directions of an element belonging considered group to a specific value for the polynomial order. A sketch of the local directions and polynomial order definition is given in Fig. 5.23. In particular, following the outcomes of the analyses previously performed on isolated panels, the starting order has been chosen to be equal to two, and it remain fixed to this value for the “short” edges, whereas on the long edges it can increase from two to seven.

Once the input file has been set up, the adaptive analysis can start. The polynomial order of the shape functions is modified on each element independently and the analysis stops when the maximum polynomial order has been reached, in the present case equal to seven. In order to compare the result with those obtained running a traditional analysis performed increasing the number of elements, different shell models for the same wing have been generated, following the same mesh strategy used for the solid models. A starting element size is initially defined, and this size is halved at
The results for the convergence analysis are reported in Figs. 5.24 and 5.25. These figures show the behavior of the first four lower frequencies computed versus the number of degrees of freedom of the models, for both the solid and the shell models. The modal shapes associated to the four frequencies are shown in Fig. 5.26.

Figures 5.24 and 5.25 show that the solid model attains a converged result requiring a number of degrees of freedom that is less than the shell model. More important, it must be remarked that even with the lowest polynomial order chosen for the shape functions (second order), the results obtained for the solid model are very close to the converged solution. This closeness is quantified in Figs. 5.27 and 5.28, where instead of the frequency computed, the relative error of the frequency with respect to the converged value is reported. As can be seen, the frequencies determined with the solid model are within 1% from the converged value.
converged solution already for the coarsest solid mesh used, whereas the shell model solution requires about four computations (therefore four mesh generations and four postprocessing operations) to attain the same accuracy in the solution. The solid model is at this stage outperforming the shell models. Some remark should be done on the difference between the shell and the solid models convergence histories: the shell model attains convergence from lower frequencies, whereas this behavior is not expected from a theoretical point of view. Again it must be remarked that the shell elements available in Nastran have no curvature and this could have an effect on the convergence behavior. This explanation is also supported by the results obtained for isolated panels shown in Chap. 2. In presence of curvature shell elements showed a non monotonic convergence behavior and in one case a convergence from below. In the case of a complete wing, apart from the curvature effects, there should be also an effect due to the orthogonal connections between elements. Considered two flat plate elements attached to each other at a 90 degree angle, there is the problem that an in-plane degree of

Figure 5.25: Convergence analysis on the third and fourth lower frequencies computed for the Straight Wing.
freedom for one elements corresponds to the “drilling” degree of freedom for the other element. The Nastran solver handles this issue in a manner transparent to the user and this could change the convergence behavior of the model, especially when there are contiguous elements at an angle close to 90 degrees or at an angle close to 0 degrees. Moreover, the figures show a difference between the converged results of the shell and solid models, nevertheless very small (below 1%). This can be imputed to some differences in the geometry of the models. For instance differences appear when a solid model is generated starting from a set surfaces by applying a thickness distribution. This can be shown comparing the trailing edge and the leading edge solid geometry shown in Fig. 5.37. The trailing edge has “more volume” than the leading edge although the thickness given as input is the same. This is due to the fact that the wing trailing edge can have changes of curvatures, typically when a transonic profile is used, as in the present case. These changes should be eventually handled in a more efficient manner by the solid generator algorithm.

5.4.2 Blended wing body

The test case shown in the preceding section was introduced to investigate the behavior of the solid elements for a simple but “real” test case. This case did not include some of the typical features a wing can have. Indeed the solid finite
Figure 5.27: Convergence analysis on the relative error of the computed frequency from the converged value for the first lower frequency of the Straight Wing.

Elements generated in the model were simple cylindrical elements\(^2\) without any twist or skewness due to the absence of a sweep angle and built-in twist in the wing.

For this reason it has been decided to run another test taking as example a more realistic configuration. The choice has fallen on a Blended Wing Body (BWB) configuration, Fig. 5.29. This configuration has been the objective of an extensive study during a European project run at TUDelft [61]. During this project several configurations have been analysed, and among them one has been picked to be the test case for the solid analysis and modeling tool. The candidate test configuration, has a span of \(s = 38\, m\) a fuselage length of \(l = 38.6\, m\) and a sweep angle of \(\theta = 45\, deg\). All the remaining relevant parameters can be found in the report [61].

\(^2\)A cylindrical element is defined as the solid element obtained from extruding a surface along a straight line.
The present solid modelling and mesh generator tool has been used to create an example mesh that is shown in Fig. 5.30 for half Blended Wing Body. This finite element model has been generated using four wing-trunks, shown in Fig. 5.29. The four wing-trunks represent respectively the BWB fuselage (trunk no.1), the inner-wing (split into two inner-wing pieces, trunks no.2 and no.3) and the outer wing (trunk no.4, without the winglet). Each of this wing-trunks has been processed separately by the solid model generator (upper block of Fig. 5.21) which eventually generated four different files containing the grid nodes and four different files containing the points for the higher order geometry description. Subsequently, the Fortran routine for the topology generation (lower block of Fig. 5.21) generated a Nastran analysis input file by connecting the different wing-trunks and by connecting the different patches of each wing trunk, following the procedure described in section 5.3. The file has been read into Nastran and
the obtained mesh is shown in Fig. 5.30. The picture represents the state-of-the-art of the capabilities of the solid geometric tool and the solid mesh generator tool. Although the generated aircraft configuration is appealing, the structural lay-out is still simple. Indeed the solid model generator cannot handle fuselage models (containing a floor, frames, bays), or ribs made of only one of their three basic components (leading-edge rib, center-rib, trailing-edge rib), or spars which are not entirely crossing the wing-trunk between two opposite sides. In addition, the solid mesh generator tool has been validated only for structured meshes, i.e. meshes for which the number of nodes on the opposite sides of a quadrilateral surface is the same. Structured meshes require only the use of hexaedral finite elements (apart from the leading edge and the trailing edge ribs for which pentahedral elements have been used). Therefore creating a structured
Table 5.4: Parameters defining the geometrical configuration of the Blended Wing Body external wing.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Root Chord</td>
<td>cr = 18 m</td>
</tr>
<tr>
<td>Tip Chord</td>
<td>ct = 4 m</td>
</tr>
<tr>
<td>Number of ribs</td>
<td>Nr = 21</td>
</tr>
<tr>
<td>Number of spars</td>
<td>Nr = 3</td>
</tr>
<tr>
<td>Span</td>
<td>s = 21 m</td>
</tr>
<tr>
<td>Skin thickness</td>
<td>tsk = 4.01 mm</td>
</tr>
<tr>
<td>Sweep angle</td>
<td>α = 45 deg</td>
</tr>
<tr>
<td>Spar thickness</td>
<td>tsp = 20 mm</td>
</tr>
<tr>
<td>Twist angle</td>
<td>β = 1 deg</td>
</tr>
<tr>
<td>Rib thickness</td>
<td>trib = 20 mm</td>
</tr>
<tr>
<td>Wing profile</td>
<td>NACA64A010</td>
</tr>
<tr>
<td>Root Rib (3) thickness</td>
<td>tsub = 50 mm</td>
</tr>
<tr>
<td>Relative thickness</td>
<td>10%</td>
</tr>
</tbody>
</table>

A mesh for such a configuration would require the same number of nodes on the profile at the centerline of the BWB (which has a chord of 40 meters) and on the profile at the tip of the BWB (which has a chord of 4 meters). This would lead to the creation of an overly fine and oversized useless mesh at the tip of the wing which can probably cause numerical stability problems due to the use of very high aspect ratio finite elements.

A much better discretization could be obtained if a hybrid mesh could be created, i.e. a mesh that uses both hexaedral and pentahedral elements. In this way the element size can be kept close to a uniform size, avoiding numerical problems. This feature is still under development in the solid modelling tool and therefore was not ready to be applied in the present work.

Still having in mind to test the behavior of the solid elements on a realistic configuration, it has been decided to generate a wing configuration that could reproduce closely the wing geometry of the BWB, avoiding the problems shown above.

For this reason the attention has been focused only on the external wing of the BWB. The geometrical parameters describing the wing are reported in Table 5.4. Some parameters needed

![Figure 5.31](image-url): Surface model for the external wing of the Blended Wing Body, made of 75 surfaces (patches): 2 leading-edge skins (upper and lower), 2 center-wing skins (upper and lower), 2 trailing-edge skins (upper and lower), 3 spars, 22 leading-edge ribs, 22 center-wing ribs, and 22 trailing-edge ribs
to be changed with respect to the original model in order to properly design the wing and have as much as possible “clean” modes at lower frequencies, i.e. modal shapes without the appearance of local motions on top of the “global” motions. This could occur when using variable thickness on different areas of the wing or applying stringers. As the meshing tool cannot handle models having a variable thickness for the upper and lower-skins, it was decided to use a constant thickness equal to $t = 4.01 \text{ mm}$. The spars and the ribs have the same constant thickness equal to $20 \text{ mm}$. The three ribs closer to the root section of the wing have instead a thickness of $t = 50 \text{ mm}$, as at a first inspection of the modal analysis results a lower thickness resulted in the appearance rib modes at the lowest frequencies.

As already explained before, no procedure has been implemented in order to define the number of elements that can be put in the chordwise direction such as the edge-wise third order polynomial expansion for the geometry of the element results in an acceptable matching with the geometry of the wing. Therefore, three different finite element models, shown in Table 5.5 have been generated. Each mesh corresponds to a target element size specified to generate the mesh. For each of the three meshes an adaptive analysis has been performed, constraining the shape functions to have a fixed order along all the edges having the size of the thickness, and allowing the order of the shape functions on the remaining edges to run freely according to the adaptive analysis. The model has structural elements which have a thickness size of the order of tenths of millimeters, keeping the largest size of the order of the meter. Differently from the previous wing it has been decided to allow the order of the shape functions along the thickness direction to start from the third order, and, to perform also a sensitivity analysis on the polynomial choice along the thickness.

Table 5.5: Three different meshes used to perform the analysis of the Blended Wing Body external wing, and maximum sizes of the three types of finite elements making the solid meshes.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>MESH-1</th>
<th>MESH-2</th>
<th>MESH-3</th>
</tr>
</thead>
<tbody>
<tr>
<td>NLE</td>
<td>NLE=1</td>
<td>NLE=1</td>
<td>NLE=2</td>
</tr>
<tr>
<td>NCOVER</td>
<td>NCOVER=1</td>
<td>NCOVER=2</td>
<td>NCOVER=4</td>
</tr>
<tr>
<td>NBAY</td>
<td>NBAY=1</td>
<td>NBAY=1</td>
<td>NBAY=1</td>
</tr>
<tr>
<td>NSPAR</td>
<td>NSPAR=1</td>
<td>NSPAR=1</td>
<td>NSPAR=1</td>
</tr>
<tr>
<td>NTE</td>
<td>NTE=1</td>
<td>NTE=1</td>
<td>NTE=3</td>
</tr>
<tr>
<td>Max Plate-Like Element</td>
<td>6269 x 1687 x 2.01 mm</td>
<td>3142 x 1761 x 4.01 mm</td>
<td>1568 x 1679 x 4.01 mm</td>
</tr>
<tr>
<td>Max Beam-Like Element</td>
<td>6269 x 50 x 4.01 mm</td>
<td>3142 x 50 x 4.01 mm</td>
<td>1568 x 50 x 4.01 mm</td>
</tr>
<tr>
<td>Max Point-Like Element</td>
<td>4.01 x 4.01 x 4.01 mm</td>
<td>4.01 x 4.01 x 4.01 mm</td>
<td>4.01 x 4.01 x 4.01 mm</td>
</tr>
<tr>
<td>No. of elements</td>
<td>841</td>
<td>1057</td>
<td>1813</td>
</tr>
</tbody>
</table>

Material = Aluminum
Young’s modulus $E = 70 GPa$, Poisson’s ratio $\nu = .33$, Density $\rho = 2700 Kg/m^3$. 
Following the mesh strategy used for generating the solid models, shell models for the wing have been created starting from the largest target element size of the solid model and halving this size at each new computational step.

The results obtained for the convergence analysis on the first four lower frequencies of the shell and solid models are shown in Figs. 5.32, 5.33. As the results for the solids were very close to each other, only the results associated to MESH-1 and MESH-3 (named respectively *solid-plt3-1* and *solid-p993-3*) are reported. For the shell models, results computed using a consistent and a lumped mass matrix formulation are reported.

The solution for the coarsest solid mesh (*solid-plt3-1*) is computed using polynomials up to order twelve for the “long” edges, whereas along the thickness-wise edges the polynomial order is kept fixed to the third, after a sensitivity analysis has been done. The solution for the solid finer mesh (*solid-p993-3*) is computed using up to the ninth polynomial order for the “long” edges and
third order thickness-wise. The modal shapes associated to these frequencies are reported in Fig. 5.34.

At a first sight the figures reveal that the solid elements don’t show the same high convergence rate of the first test case. In particular all the solid meshes chosen have the same rate of convergence, which in this case is the same as the shell models. For better insight Figs. 5.35, 5.36 show the convergence curves for the relative error of the frequency with respect to the converged value. Still they show that the convergence rate is the same for both the shell and solid models.

Different explanations can be given for the slower convergence rate of the p-elements models:

- Presence of singularities
- “Unfamiliar” mode couplings
- Poor geometrical approximation

**Figure 5.33:** Convergence analysis on the third and fourth lower frequencies computed for the BWB Wing
Singularity analysis

For the first hypothesis other authors [23, 26, 33] showed that the p-formulation finite-elements lose their high (exponential) convergence rate when singularities appear in the problem.

Example of singularities are the presence of cracks or the application of concentrated loads for a static analysis problem. In a modal analysis problem, like the present, where there are no cracks or concentrated loads, these singularities can only come from the geometry of the model, and they can be related to the fact that now the elements have a more remarked skewness and a taper due to the presence of a sweep angle and a taper for the wing.

Skewness and taper were present also in the elements of the straight wing studied before even if the wing as such has no sweep or taper. In fact, due to the geometric process followed for the construction of a solid model, taper and skewness appear, as shown in Fig. 5.37. The different meshes created for the straight wing have been checked using the geometrical pre/post processor of Nastran. These checks showed that about one third of the total elements of the mesh overcome the default boundary values marked by Nastran as acceptable for the elements taper ratio and skewness. The same check has been performed on the BWB meshes. The number of elements overcoming the limit increases to about one half of the total number of elements present in the model, and this number stays constant for each of the three meshes created for the analysis. However it should be understood that the geometric pre/post processor of Nastran, when running these checks, does not take into account the higher order geometric representation of the finite elements. Therefore the numbers given above (one third and one half of the total number of elements for the two meshes) must be taken as just an indication. To verify if the slower convergence rate is due to an increase in the skewness and taper of the finite elements, a test has been set-up. The straight wing studied in section 5.4.1 has been modified giving the same sweep angle, taper ratio and built-in twist of the BWB external wing (therefore keeping a different span and thickness), resulting in the geometry shown in Fig. 5.39. The geometric parameters are reported in Table 5.6. A mesh corresponding to the MESH-1 parameters of Table 5.5 has been generated, and an adaptive modal analysis has been carried out using the same polynomial orders for the straight wing case. On the thickness-wise edges the polynomial order is kept fixed to the second whereas on all the other edges the polynomial orders run from the second to the seventh. The results of this analysis are presented in Fig. 5.39. The figure reports the error of the computed frequency with respect to the converged value for the mode considered, for the MESH-1 case of the straight wing studied before (name as straight) and for the modified straight wing (named straight-modified). The new mode shapes for the modified straight wing, associated to the frequencies considered, are reported in Fig. 5.40.

The figure shows that for the modified straight wing there is a negligible variation in the (high) convergence rate of the frequencies. This shows that in the case of the BWB external wing, the slow convergence rate cannot be imputed to the presence of the taper and sweep angle in the wing. These results proof
Figure 5.34: Modal shapes of the first four lower modes for the wing in Fig. 5.31.

Table 5.6: Parameters defining the geometrical configuration of the modified straight wing shown in Fig. 5.38.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Root Chord</td>
<td>$c_r = 1\text{ m}$</td>
</tr>
<tr>
<td>Tip Chord</td>
<td>$c_t = 0.25\text{ m}$</td>
</tr>
<tr>
<td>Span</td>
<td>$s = 10\text{ m}$</td>
</tr>
<tr>
<td>Skin thickness</td>
<td>$t_{sk} = 2.01\text{ mm}$</td>
</tr>
<tr>
<td>Spar thickness</td>
<td>$t_{sp} = 2.01\text{ mm}$</td>
</tr>
<tr>
<td>Rib thickness</td>
<td>$t_{rib} = 2.01\text{ mm}$</td>
</tr>
</tbody>
</table>

“Unfamiliar” mode couplings

Although the modified straight wing has many geometrical properties similar to the BWB wing, the output in terms of modal shapes is quite different. Indeed, the mode shapes of the modified straight wing reported in Fig. 5.34 show that the torsional and the flexural modes are uncoupled, whereas the mode shapes for the BWB wing reported in Fig. 5.34 show that there is a strong coupling between the torsional and the bending modes. The slower convergence rate exhibited by the solid $p$-elements in the analysis of the BWB could be imputed to the difficulty of capturing this coupling. Therefore it is useful to investigate this aspect too. One of the modes of the modified straight wing revealed a slight coupling between bending and torsion. This mode resembles one of the modes
of the BWB wing. The mode shape is shown in Fig. 5.31. From the figure the coupling is not evident but it becomes clearer when an animation of the modal shape is done. In the same figure the convergence analysis of the relative error for the fifth mode frequency is performed. It was expected that in the process of solving the modal problem, giving more energy to this mode could give perhaps a better insight on the role of the bending-torsion coupling. It was not clear how to assign more energy to this specific mode, therefore it has been decided to run the same modal analysis lowering the Young’s modulus. Making the structure more flexible, gives more energy to all the modes of the structure and not only to the one of interest. If the displacement field is expanded as a linear combination of the mode shapes, a lower flexibility has the effect of increasing the number of modes needed to represent accurately the displacement field, and in this sense it was intended to emphasize the role of the specific mode under consideration, therefore making it “more visible”. The resulting mode shape is shown in Fig. 5.32a. The coupling in this case is more remarkable, but again it is
Figure 5.36: Convergence analysis on the error of the computed frequency relative to the converged value for the third and fourth lower frequency of the BWB Wing.

Figure 5.37: Distorted elements in the section of the Straight Wing.

more visible when the mode is animated. A convergence analysis on the relative error of the computed frequency has been performed on this mode shape. The results, presented in Fig. 5.32, show that the convergence rate is as high as for the case of the uncoupled modes of the straight wing. From this it can be
concluded that a strong coupling between torsional and bending modes does not affect the high convergence rate capabilities for the solid p-elements, and another effect that can explain the slow convergence has to be found.

**Geometrical approximation**

Another source of problems for the convergence analysis of the solid p-elements can be a bad approximation of the geometry. The analyses that have been performed on the BWB external wing used three different meshes without making any study on the possibility of the third order polynomial reconstruction to match the actual geometry of the edges of the different wing parts. In particular the coarsest mesh used for the BWB has the minimum number of elements possible, where each finite element corresponds to the parts of the wing between two
consecutive connections/intersections with other structural elements. The mesh
is therefore designed by the distributions of the different structural elements in
the wing and their connections and not by the geometrical approximation. The
results of Figs. 5.32, 5.33 show that the geometrical approximation given by the
meshes used is good enough. Indeed the two meshes represented, MESH-1 and
MESH-3 of Table 5.5, tend to the same converged value, and the MESH-3 has
a number of elements that is double the number of elements of MESH-1. If the
geometry was poorly represented, there would have been a remarkable dif-
ference between the two meshes. Instead the two meshes give the same con-
vergence rate and tend to the same asymptotic value for the frequency. There-
fore there is no evidence for a problem with the geometry approximation.
5.4. Applications

(a) Fifth mode shape.

(b) Convergence of the relative error for the fifth frequency.

Figure 5.42: Mode shape and convergence analysis of the relative error for the fifth mode of the weakened modified straight wing

Discussion

The analysis given so-far failed in explaining why the convergence rate of the solid models degrades so remarkably in the case of the BWB external wing. Nevertheless, the motivation for this behavior has been found. Indeed it is sufficient to take a look at the mode shapes reported in Fig. 5.34. In the figure the modal displacements are reported only at the grid nodes (8 per finite element) without taking into account the internal degrees of freedom that are added during the adaptive analysis. The Nastran pre/post processor offers the possibility to give output information on a grid of 64 points for each (p-formulation) finite element, in order to take into account the presence of internal degrees of freedom. Following this procedure the four modes of the BWB external wing are reported in Fig. 5.43 with more degrees of freedom. It can immediately be noted that the previous pictures missed to represent the modal displacements that appear on the panels comprised between two ribs. This is the difference between the behavior of the BWB wing and the behavior of the straight wing (and the modified straight wing) shown before. The straight wings don’t have such huge displacements at panel level. This difference explains the reduction in convergence rate for the solid and the shell models built for the BWB wing. Indeed with the increase of the number of degrees of freedom the local oscillations start to be captured, and in order to be correctly resolved, more degrees of freedom are needed. Therefore the convergence rate keeps on being “pre-asymptotical” for a large part of the whole computational time. In addition the much lower local flexibility of the BWB wing on few panels makes the stiffness matrix of the problem ill-conditioned and almost singular. This statement can be easily proven creating a weaker wing by reducing the skin thickness and/or reducing the number of ribs. In such a situation the Nastran solver stops after the first or second adaptive steps because of singularity occurring in the stiffness matrix. This cannot be considered as a numerical drawback connected to the use of p-elements, also because the shell models exhibit the same convergence/singularity problems. Instead it is something that can be exploited in case an automated


analysis is performed. The real issue is that from the design point of view, the structural lay-out of the BWB external wing used as test case is not acceptable, because it is too weak. The panel displacements are of the same order of magnitude as the wing tips displacements. Large local displacements appear already on the first lower frequency modal shapes. Such a wing cannot be accepted in a design unless actions are taken to increase the local stiffnesses where needed. While running an adaptive analysis using p-elements, a slow convergence rate can be used as a warning that the design of the wing is not acceptable from the vibration point of view.

As an appendix, Table 5.7 reports the CPU time for each computation of both shell and solid p-elements models. It can be seen that the first four computations for the shell models require less time than the first computation of the solid model. The CPU time seems to be a drawback of the p-formulation approach. Nevertheless it must be considered that each successive computation of the shell model requires a new mesh generation, which if not automatically performed requires a lot of time to be performed, whereas for the solid p-elements model only one mesh is needed. Besides it must be taken into account that for the solid model (when the wing is correctly designed as the straight wing) after a couple of runs the solution is converged within engineering acceptance. Moreover comparing CPU times is not fully trustworthy. Indeed these times were taken from a Nastran output file, which does not specify how the times are computed, and it must be considered that the computations were performed on a non-standalone machine. Finally it must be remarked that the availability of parallel solvers and cheap computational power can ease dramatically the problems connected with the CPU times of the solid p-elements.

![First mode shape](image1)
![Second mode shape](image2)
![Third mode shape](image3)
![Fourth mode shape](image4)

Figure 5.43: Mode shapes for the BWB external wing represented taking into account the internal degrees of freedom of the solid p-elements.
5.5 Conclusions and recommendations

<table>
<thead>
<tr>
<th>Run no.</th>
<th>Shell model Cpu time (s)</th>
<th>Solid model Cpu time (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.4</td>
<td>61.1</td>
</tr>
<tr>
<td>2</td>
<td>2.5</td>
<td>311</td>
</tr>
<tr>
<td>3</td>
<td>4.6</td>
<td>768</td>
</tr>
<tr>
<td>4</td>
<td>27.1</td>
<td>1701</td>
</tr>
<tr>
<td>5</td>
<td>130.6</td>
<td>1897</td>
</tr>
<tr>
<td>6</td>
<td>893</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>2171</td>
<td></td>
</tr>
</tbody>
</table>

Table 5.7: Comparison of the CPU times for the shell and solid p-elements models of the BWB wing.

5.5 Conclusions and recommendations

The results shown so-far on the modal analysis of wing-trunks having different geometric configurations have proven the valuability, flexibility and robustness of the solid p-elements in performing the modal analysis of wing-trunks. The solid modelling tool, created to support the analysis using solid elements, has shown its flexibility in changing the wing solid geometry configuration and providing different geometries.

The solid p-elements have proven their robustness and accuracy in the modal analysis of wing-trunks using coarse meshes made of elements (shell-like, beam-like, point-like elements) having extreme aspect ratios. The results were very accurate (practically converged, from an engineering point of view) even using the most coarse mesh possible, it was just sufficient to generate a solid model where all the connections between the different solid patches were found and directly translated into a finite element model, thus avoiding the “standard” meshing process.

The possibility offered by the solids to model any kind of structural element coupled with the adaptive analysis capability overcome the restrictions of the traditional approach based on the use of linear shell finite-element types, and makes it a naturally suitable tool to support a KBE approach for the structural and aeroelastic design of aerospace vehicles.
Chapter 6

System Identification for Flutter Analysis

6.1 Introduction

In Chapter 1.2 of the present work, Fig. 1.3 shows the components of an aeroelastic analysis tool, which in the simplest case could be seen as made of a structural block and an aerodynamic block. Following the traditional approach of expanding the displacement field (output of the structural block) in terms of the vibration modes of the structure, in the preceding chapters it was presented an approach to determine the modes in a way suitable to be embedded in an automatic tool.

With respect to the aerodynamic block, as already stated before, if the aerodynamic operator is linear(-ized), the modal expansion for the displacements leads to the definition of the Generalized Aerodynamic Force (GAF) matrix. This matrix is the transfer function between the generalized forces and the modal displacements. The use of modal techniques results in computationally efficient models, which are widely used for performing linear flutter analysis.

For transonic flows, the GAF matrix becomes a function of the steady condition of interest (defined by the freestream’s Mach number and the angle of attack) due to the presence of shocks. The correct prediction of the fluid flow steady state properties, i.e. shock location and strength, is important for the linearization which eventually results in the GAF matrix. Methods like the Transonic Doublet Lattice or the Boundary Element method (defined also Field Panel method when transonic flows are considered) are available to derive the GAF matrix in transonic conditions. The applicability of these methods is limited by the assumptions on which they are based, i.e. weak shocks. In addition, the GAF matrix as such can be used only for the identification of the stability margins of the system. A wider applicability, for example the design of a flutter suppression system or a gust alleviation control system, requires the application of techniques such as the Rational Fraction approximation which reduces the number of states of the aerodynamic system, i.e. the poles of the GAF matrix,
to a finite number, providing a specific ROM for the aerodynamics block.

The constraints of the above mentioned ROM for the linearized analysis methods could be overcome using a CFD code. A CFD code can analyse any type of shock strength – the fluid flow static non-linearity –, as well as any type of perturbation around the shock – small or large – thus allowing the analysis of either a linearized or a non-linear flow model around the steady-state. Following the traditional approach for performing stability analysis, first a GAF matrix could be determined linearizing the CFD responses around a steady configuration, and then a ROM could be derived using a finite state approximation technique. An alternative approach is the one offered by system identification techniques applied to the CFD code. Feeding the CFD code, treated as a black-box system, by suitable inputs and recording the outputs, a ROM could be derived directly by CFD simulations.

Over the last years significant effort has been spent on the development of Reduced Order Models (ROMs) of the Unsteady Aerodynamic operator, both in frequency and in time domain. Some examples of frequency domain ROMs are the indicial responses by Balhaus [67], the pulse transfer function analysis by Lee-Rausch [68], field panel method by Fokin [69], Chen [70], Iemma [71], the Volterra theory by Silva [72], Marzocca [73], Ravich [74] and the Proper Orthogonal Decomposition (POD) by Silva [75], Thomas [76]. Examples of time domain ROMs are those based on the ARMA model by Cowan [77], the Volterra theory by Silva [78], the unit sample responses from Gaitonde [79] and the Proper Orthogonal Decomposition by Lucia [80] and Beran [81].

Volterra theory and Proper Orthogonal Decomposition ROMs have received particular attention, as they are applicable to non-linear systems, whereas the other ROMs are based on linear system representations. The ROMs are intended to represent the dynamics of the (aerodynamic) system around the non-linear steady state (transonic) flow-condition. Such dynamics can be either linear or non-linear. Silva [72] showed for a NACA0012 profile case the difference in modeling the dynamics as pure linear or linearized from a non-linear flow condition.

Nevertheless the boundary between modeling the dynamics using a linear or a non-linear model is not clear. None of the papers mentioned above quantifies the weight of the non-linear terms in the flow equations or defines clearly the motivation to adopt a linear(-ized) model. They just assume that the non-linear terms have small contributions by considering small amplitudes of the modal shapes in the system identification procedure. Only Dowell [66] writes that if the non-linear steady flow field, including the static shock strength and location, is accurately captured, the dynamic perturbations about the steady flow can be studied using linear models. In any case the results show good agreement with experimental data, in most cases represented by the flutter plot of the well known AGARD 445.6 wing [82].

The objective of the present chapter is to investigate whether a ROM suitable for aeroelastic analysis in a preliminary design environment, can be derived using a linear system identification procedure. Evidence has been shown for a particular case [83], that even using an Euler code, the aerodynamic system can behave linearly, independent of the amplitude of the input used in the system identifica-
tion cycle, thus avoiding the use of a non-linear system identification procedure. In terms of computational costs the possibility of avoiding a non-linear system identification procedure is appealing. For example a Volterra ROM requires that each mode considered is excited separately. Instead, for a linear ROM a Multi-Input Multi-Output (MIMO) identification procedure, requiring only one flow solution, can be used [77]. Besides, in a Volterra based ROM, more than one CFD run is required for each mode considered. The number of simulations to be performed is depending on the accuracy of the non-linear approximation.

In the following sections, the influence of the non-linear terms on the dynamical behavior of a profile in a transonic flow will be investigated. This is done by providing the aerodynamic system with suitable inputs, like impulses, having different amplitudes. The computed responses are normalized to the smallest input amplitude. Comparison of these normalized responses together with their frequency content provides qualitative and quantitative information about the non-linear behavior of the system.

### 6.2 NACA64A010 test case

The first test case is based on the NACA64A010 profile studied in the AGARD report No. 702, case CT6 [84]. This profile has been largely studied in literature. Its importance stems from the fact that when attached to a plunge-pitch spring system, with the structural parameters of the ISOGAI case [85, 86], it exhibits inviscid Limit Cycle Oscillations phenomenon. For this phenomenon the aerodynamic non-linearities play a fundamental role.

In the AGARD report, case CT6, the profile is set at an angle of attack \( \alpha = -0.21 \) degrees and Mach number \( M_{\infty} = 0.796 \). Experimental results (Pressure, Lift and Pitch Moment coefficients) are reported for both a steady analysis and an oscillatory analysis around the quarter chord of the profile. These results are used as validation data for a computational numerical model built using the commercial package Fluent 6.1.22 [87], based on a finite volume solver of the Euler equations. A second order upwind, implicit scheme has been chosen. The computational domain is circular with a radius which is 40 times the profile chord-length. A structured, \( 80 \times 50 \) mesh (80 elements on the upper and lower parts of the airfoil, 50 elements radial-wise) has been chosen after a convergence study on the mesh size.

Fig. 6.1 shows the comparison between the numerical and the experimental Pressure Coefficient \( (C_p) \) on the upper and lower part of the profile for the steady analysis performed at \( \alpha = -0.21 \) degrees and \( M_{\infty} = 0.796 \). The Mach number at the shock, Fig. 6.2, is less than 1.2 and the corresponding entropy jump, Fig. 6.3, is negligible.

Figures 6.4 and 6.5 report the Lift \( (c_l) \) and Pitch Moment \( (c_m) \) coefficients responses versus angle of attack for a sinusoidal oscillation of the profile around its quarter chord, at a reduced frequency of \( k_r = 0.202 \) \( (k_r = \omega b/U_{\infty}) \) with amplitude of \( \Delta \alpha = 1.01 \) degrees. The numerically determined \( c_l \) agrees quite

\[ \omega \text{ is the circular frequency, } b \text{ is the semichord and } U_{\infty} \text{ is the freestream flow velocity} \]
well with the experimental one whereas the $c_m$ shows a certain difference. This difference, detected also by other authors \cite{88}, is due to the inability of the Euler solver to correctly capture the dynamics of the shock. Indeed the shock motion is affected also by viscous effects that are not modeled in this case. Nevertheless, the quality of the phenomenon, i.e. shock position and boundaries of the $c_m$, are correctly captured. The dynamic analysis presented above does not say anything about the expected non-linear behavior of the system and the importance of the non-linear terms of the Euler equation to the response of the system. For improved insight, the frequency content of the output is evaluated calculating the Fourier transform of the time-histories for the $c_l$ and $c_m$ perturbations. The plots, Figs. 6.6 and 6.7, show a dominant peak in correspondence of the frequency at which the system has been excited (reduced frequency $k_r = \omega_b/U_\infty = 0.202$ corresponding to a frequency $f = 17.41$ Hz for the undisturbed conditions chosen). More visible on the $c_m$ plot, peaks with decreasing amplitudes appear at
6.2. NACA64A010 test case

Figure 6.3: Variation of the entropy jump $S_2 - S_1$ (normalized with the universal gas constant $R$) across a normal shock, with the Mach number ($M_1$) before the shock.

Figure 6.4: Lift Coefficient versus angle of attack for a sinusoidal oscillation ($k_r = 0.202$, $\Delta \alpha = 1.01$ degrees), NACA64A010 profile, pitch motion.

frequencies that are a multiple integer of the excitation frequency. The higher frequencies show the contributions of the non-linear terms in the Euler equations, and they are almost two orders of magnitudes smaller than the contribution of the input frequency. Of course their influence is expected to grow as the angle of attack around which the analysis is carried on increases, i.e. as the shock strength increases. In order to have an idea of the non-linear effects contribution, a steady analysis has been performed at an increased angle of attack of 3 degrees. The shock in this case, Fig. 6.8, is much stronger. The Mach number before the shock is $M_1 = 1.4$ and the entropy jump is one order of magnitude
Figure 6.5: Moment Coefficient versus angle of attack for a sinusoidal oscillation \((k_r = 0.202, \Delta \alpha = 1.01 \text{ degrees})\), NACA64A010 profile, pitch motion.

Figure 6.6: Modulus of the Fourier transform for the perturbation of the \(c_l\) \((k_r = 0.202, \Delta \alpha = 1.01 \text{ degrees})\), NACA64A010 profile, pitch motion.

bigger than the previous case at \(\alpha = -0.21\) degrees. The entropy variation extends through the shock line and it is convected downstream, as shown in Fig. 6.9. In the previous case the entropy variation is hardly noticeable since it is limited to the root of the shock (for this reason, no contour plot is shown). The flow condition shown by Fig. 6.9 is at the limit of the theoretical capability of the Euler formulation since, in reality, shock-boundary layer interactions can occur and they are not captured by this formulation. Another oscillatory analysis has been done, with an oscillation amplitude of 1 degree around the profile quarter chord. In this case, Figs. 6.10 and 6.11 the Fourier transform of the \(c_l\) and
6.2. NACA64A010 test case

\[ \left| \hat{c}_m \right| \]

\[ \text{Frequency (Hz)} \]

\[ \text{Cm} \]

\[ \text{Figure 6.7: Modulus of the Fourier transform for the perturbation of the } c_m \]
\[ (k_r = 0.202, \Delta \alpha = 1.01 \text{ degrees}, \text{NACA64A010 profile, pitch motion}. \]

\[ \text{Mach number} \]

\[ \text{x/c} \]

\[ \text{Figure 6.8: Mach number over the NACA64A010 profile for a steady angle of attack of 3 degrees.} \]

\[ c_m \] show obviously an increased contribution of the non-linear terms. However there is one order of magnitude difference between the higher harmonics and the exciting frequency. The results presented so far show that when the system is excited at one frequency, additional frequencies appear due to the non-linear behavior of the system. The low amplitudes of these harmonics (difference of order of magnitudes) make the contribution of these additional frequencies negligible and the response of the system can be well represented by the harmonic corresponding to the input frequency.

As an appendix to the analysis shown above, it is interesting to remark that
the profile oscillations result in a shock motion. The amplitude of the shock motion is directly related to the frequency of the oscillations. With respect to this motion an aerodynamic system behaves like a low band filter with the result that the shock is “insensitive” to high frequency oscillations. As an example, different harmonic analyses have been performed varying the frequency of the first test case (angle of attack $\alpha = -0.21$ degrees, $\Delta \alpha = 1.01$ degrees) and measuring the amplitude of the shock motion, normalized to the unit chord. Results are presented in Table 6.1. The amplitude of the reference shock motion, at a reduced frequency $k_r = 0.202$ is 9.02%. The table shows that this amplitude almost doubles by scaling the excitation frequency to one-third of the reference frequency and lowers to almost one-third of the reference amplitude motion when doubling the excitation frequency. The numbers reported clearly show that the dependence of the excitation frequency is non-linear. When the aerodynamic system is excited with a large-bandwidth signal the dependence of the shock motion on the excitation frequency is expected to be less clear. Indeed, due to the system’s non-linear behavior, additional (higher) frequencies appear in the system’s response as shown in Figs. 6.10, 6.11. These frequencies can couple and have a non-negligible effect on the system response, and the correlation between high frequency excitation, small amplitude shock motion and linearization of the flow equation around the steady state can be lost.

In order to give an insight on the non-linear effects role over the (aerodynamic) system behavior, the same oscillatory analysis is performed using a typical input employed in system identification, consisting of a Gaussian shaped impulse called “enlarged impulse”, Fig. 6.12. In the present case the time law for the

<table>
<thead>
<tr>
<th>Excitation frequency</th>
<th>Shock motion amplitude</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k_1 = 0.202$</td>
<td>9.02%</td>
</tr>
<tr>
<td>$k_2 = 1/2k_1$</td>
<td>9.21%</td>
</tr>
<tr>
<td>$k_3 = 2k_1$</td>
<td>3.35%</td>
</tr>
<tr>
<td>$k_4 = 1/3k_1$</td>
<td>16.2%</td>
</tr>
</tbody>
</table>

Table 6.1: Variation of the shock motion amplitude with the excitation frequency.
6.2. NACA64A010 test case

Figure 6.10: Modulus of the Fourier transform for the perturbation of the $c_l$ ($k_r = 0.202$, $\Delta\alpha = 2.0$ degrees), NACA64A010 profile, pitch motion.

Figure 6.11: Modulus of the Fourier transform for the perturbation of the $c_m$ ($k_r = 0.202$, $\Delta\alpha = 2.0$ degrees), NACA64A010 profile, pitch motion.

The perturbation in the angle of attack $\alpha$ is defined by:

$$\alpha(t) = Amp \cdot e^{-\left(\frac{t - T_{imp}^2}{K \cdot T_{imp}}\right)^2}, \quad (6.1)$$

where $T_{imp}$ is the time interval during which the function is not zero, $Amp$ is the amplitude of the input function, $K$ a factor that determines the smoothness of the derivatives of the input signal at the time 0 and $T_{imp}$ [89]. In the present
work the parameters have been chosen such that the bandwidth equals a reduced frequency of $k_r = 1$, Fig. 6.13, resulting in $K = 0.08$ and $T_{imp} = 0.027$ s. A better insight on the procedure used to define all the parameters for the excitation of the system and the relation to the physical parameters of the problem, such as Mach number and frequency bandwidth, can be found in reference [90] or in appendix A. Again a pitch motion, with time history defined by the enlarged impulse, has been assigned to the profile set at an angle of attack of $\alpha = -0.21$ degrees. Two different pitch amplitudes have been selected for the

---

**Figure 6.12:** Time-law used for the pitch-angle ($\alpha$) excitation, normalized to amplitude $A_{mp} = 1$.

**Figure 6.13:** Modulus of Fourier transform of the pitch-angle ($\alpha$) excitation.
6.2. NACA64A010 test case

Figure 6.14: Scaled time histories of $c_l$ for different perturbations around 0.21 degree steady angle of attack, NACA64A010 profile, pitch motion.

Figure 6.15: Scaled time histories of $c_m$ for different perturbations around 0.21 degree steady angle of attack, NACA64A010 profile, pitch motion.

For each simulation, the time histories of the $c_l$ and $c_m$ as well as the shock movement have been recorded. A linearity check of the system’s behavior has been performed normalizing the responses to the smallest amplitude of the input signals: in Figs. 6.14 and 6.15 the scaled time-histories of the $c_l$ and the $c_m$ are

analysis, $Amp_1 = 0.21$ degrees and $Amp_2 = 1.05$ degrees. These amplitudes correspond respectively to a perturbation of 100% and 500% of the steady angle of attack, and they are both far from the assumption of “small” perturbations.
reported, limited to a time-range where the transient response is appreciable. As can be seen, the two curves lie on top of each other. A better insight on the system’s behavior is gained performing a frequency domain analysis. Thus for each time-history a Fourier transform has been calculated and divided by the Fourier transform of the corresponding input, frequency by frequency. For a linear system, this procedure provides the transfer function of the system, which is independent from the amplitude of the input signal. In particular, for an aerodynamic linear system, this gives the contribution of the pitch mode to the GAF matrix. For a non-linear system, as the present case, this approach can only quantify the magnitude of the non-linearities, as the scaled responses depend on the input’s amplitude. Figures 6.16 and 6.17 show that the scaled responses overlap for each frequency of the bandwidth of interest, thus the system behaves linearly. In this case the shock measured oscillation is far below 1% of the chord length, and its strength is defined by the steady flow condition, Fig. 6.21, with negligible entropy loss. For this particular shock strength/movement combination a linear model for the dynamics of the system is appropriate. For a complete understanding, the procedure performed above for the pitch motion has been applied also for the plunge mode. The plunge motion is assigned in terms of (vertical) velocity. The amplitude of this motion is taken equal to the maximum velocity reached by the profile during the pitch motion (i.e. vertical velocity at the trailing edge). This way, the perturbations of the plunge and pitch modes are comparable. Figs. 6.18 and 6.19 report the Fourier transform of the $c_l$ and $c_m$ for the plunge motion, divided by the Fourier transform of the corresponding input, frequency by frequency. In this case some differences are evidenced, especially for the real part of the $c_l$. For the $c_m$ the differences are less evident, and this is unexpected as the $c_m$ is more sensitive to the shock motion. Nevertheless, although differences are more evident than the other cases, it should be considered that a 500% perturbation is a quite extreme excitation which has been used just to mark clearly the differences in the $c_m$ normalized responses for the pitch excitation case.

To check the validity range of the system’s linearity, it is useful to carry the previous analysis around a condition in which the shock strength is increased. For this purpose the analysis is performed about 2 degrees angle of attack. In this condition the shock is strong, and again a shock boundary-layer interaction is a physical possibility that the Euler solution cannot capture. Figures 6.20 and 6.21 display the scaled time-histories of $c_l$ and $c_m$ for 1%, 25% and 50% perturbation of the steady angle of attack. The responses show more differences than the 0.21 degree angle of attack case, but still they are not clearly noticeable. Again the differences are more appreciable if the Fourier transform is calculated. Figures 6.22 and 6.23 show the real and imaginary parts of the Fourier transform of the $c_l$ and $c_m$ divided by the Fourier transform of their respective input. Differences between the various input amplitudes can be noticed especially for

---


dFrom now on, the scaled responses will be referred as components of the GAF matrix. It should be clear that this is strictly true only when the responses are linearized around a steady condition.
the $c_m$. In this case the shock movement ranges from values between 0.6% and 1.5% of the chord with a slight change in its strength compared to the steady value. The differences between the curves are such that the dynamics of the (aerodynamic) system can be considered linear. For a linear system the quantities calculated are the contribution of the pitch mode to the Generalized Aerodynamic Force matrix, which consists also of the generalized forces due to the plunge mode of the profile. Figures 6.24 and 6.25 show again that the system
response to the plunge motion is almost independent from the magnitude of the input, with the non-linear effects even less evident than the pitch motion case.

The components of the GAF matrix have been determined with the procedure shown above, dividing frequency by frequency the Fourier transforms of the output and the input of the CFD code. A good check on the quality of this particular system identification method is to compare the GAF matrix components at zero reduced frequency \( (k = 0) \) with stability derivatives. In particular.
a comparison of the lift coefficient due to pitch motion GAF component (shown in Figs. 6.18 and 6.22) has been done with the slope of the lift coefficient versus angle of attack curve, which can be obtained running steady analyses at different angles of attack. Similarly the aerodynamic moment coefficient due to pitch GAF component (shown in Figs. 6.17 and 6.23) has been compared with the local slope of the aerodynamic moment coefficient versus angle of attack. The $c_l$ and $c_m$ versus angle of attack curves are reported in Figs. 6.26 and 6.27. The
$c_l$-$\alpha$ curve is linear up to 2 degrees, whereas for higher angles the slope changes. This change can be explained with the presence of a strong shock already at 2 degree angle of attack, with non-zero entropy gradients after the shock. These gradients make the flow rotational and therefore separation can occur, resulting in a slope change for the $c_l$-$\alpha$ curve.

The $c_m$-$\alpha$ curve instead is linear between $-1$ to $+1$ degrees. For angle of attacks higher than $+1$ degree the slope start increasing. This behavior can be explained...
6.2. NACA64A010 test case

Figure 6.24: Fourier transforms for the scaled time histories of $c_l$ for different perturbations around 2 degree steady angle of attack, NACA64A010 profile, plunge motion.

Figure 6.25: Fourier transforms for the scaled time histories of $c_m$ for different perturbations around 2 degree steady angle of attack, NACA64A010 profile, plunge motion.

with the summing up of effects linked to the shock strength (non-zero entropy gradients after the shock) and the change of the shock position with the angle of attack which significantly affects the aerodynamic moment. Using finite differences the slopes of the $c_m$-$\alpha$ and $c_l$-$\alpha$ curves are determined at $-0.21$ and $2$ degrees angle of attack and compared with the corresponding values at zero reduced frequency ($k = 0$) that can be read from Figs. 6.16, 6.22, 6.17, 6.23. This comparison is reported in Table 6.2. Results show that the steady values differ
from the unsteady values by few percent, with an increasing difference as the angle of attack grows. This rising difference can be explained with the increase in the shock strength at higher angles and the higher sensitivity of the results to the discretization errors that can occur in the CFD simulations. In addition, it must be considered that the values determined at $k = 0$ in the unsteady case are extrapolated from the results determined at the lowest frequency sampled during the identification process. Nevertheless the differences between the steady and the unsteady results are such that the identification procedure can be considered satisfactorily accurate. The analyses and results presented so far have been applied to one profile only. The linear behavior shown in the results cannot be considered to be general behavior of all profiles. In order to get a more general understanding, the same linearity check has been performed on two other different profiles for the pitch motion case, which is the most important for the appearance of the non-linear effects.
6.3 MBBA3 test case

Table 6.2: Comparison between steady computations and zero reduced frequency GAF matrix values (unsteady computations) for the lift-coefficient-due-to-pitch curve ($c_l$-α) slope and the aerodynamic moment-coefficient-due-to-pitch curve ($c_m$-α) slope.

<table>
<thead>
<tr>
<th></th>
<th>$\partial c_l / \partial \alpha$</th>
<th>$\partial c_m / \partial \alpha$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c_l$-α curve slope at 0.21 degrees</td>
<td>12.29</td>
<td></td>
</tr>
<tr>
<td>Steady</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Unsteady (input $\Delta \alpha$% = 100%, 500%)</td>
<td>12.37 12.35</td>
<td></td>
</tr>
<tr>
<td>Relative difference</td>
<td>0.65 % 0.49 %</td>
<td></td>
</tr>
<tr>
<td>$c_m$-α curve slope at 0.21 degrees</td>
<td>$\partial c_m / \partial \alpha$</td>
<td></td>
</tr>
<tr>
<td>Steady</td>
<td>0.638</td>
<td></td>
</tr>
<tr>
<td>Unsteady (input $\Delta \alpha$% = 100%, 500%)</td>
<td>0.647 0.639</td>
<td></td>
</tr>
<tr>
<td>Relative difference</td>
<td>1.41 % 0.16 %</td>
<td></td>
</tr>
<tr>
<td>$c_l$-α curve slope at 2 degrees</td>
<td>12.35</td>
<td></td>
</tr>
<tr>
<td>Steady</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Unsteady (input $\Delta \alpha$% = 1%, 25%, 50%)</td>
<td>12.82 12.64 12.62</td>
<td></td>
</tr>
<tr>
<td>Relative difference</td>
<td>3.8% 2.35% 2.19%</td>
<td></td>
</tr>
<tr>
<td>$c_m$-α curve slope at 2 degrees</td>
<td>$\partial c_m / \partial \alpha$</td>
<td></td>
</tr>
<tr>
<td>Steady</td>
<td>1.62</td>
<td></td>
</tr>
<tr>
<td>Unsteady (input $\Delta \alpha$% = 1%, 25%, 50%)</td>
<td>1.69 1.65 1.68</td>
<td></td>
</tr>
<tr>
<td>Relative difference</td>
<td>4.32% 1.85% 3.7%</td>
<td></td>
</tr>
</tbody>
</table>

6.3 MBBA3 test case

The MBBA3 profile is a supercritical profile with a maximum thickness of 8.9%, shockfree for the design conditions of $M_\infty = 0.76$, $\alpha = 1.3$ degrees. The analysis has been performed taking a steady condition with the same design Mach number and an off-design steady angle of attack of 2 degrees. In this case the Mach number at the shock reaches the maximum value of about $M_1 = 1.4$. A linearity check has been performed assigning two inputs: a “small” amplitude input, equal to 1% of the steady angle of attack and a “large” amplitude input equal to 25% of the same angle. Figures 6.29 and 6.30 show the Fourier transforms for the scaled responses of the $c_l$ and $c_m$. Negligible differences can be noticed between the different responses, therefore the dynamics of the profile can be considered linear. The results agree with those found in reference [91].

6.4 RAE2822 test case

The RAE2822 profile is a subcritical profile with a maximum thickness of 12.1%. Also for this profile a linearity check has been performed around a steady angle of attack of 2 degrees, condition at which a strong shock (Mach number at the shock around 1.4) appears. A pitch motion has been assigned using the same amplitudes of the MBBA3 profile. Figures 6.32 and 6.33 show again that the dynamics around the steady condition is linear.
Figure 6.28: Mach number along the MBB profile at 2 degree angle of attack.

Figure 6.29: Fourier transforms for the scaled time histories of $c_l$ for different perturbations around 2 degree steady angle of attack, MBBA3 profile, pitch motion.

### 6.5 Flutter results

A first flutter analysis with the V-g method has been performed for the NACA64A010 airfoil, at an angle of attack of $-0.21$ degrees, using the components of the GAF matrix shown in Figs. 6.14-6.19. The profile is restrained in bending and torsional motion by two springs simulating the stiffness of the wing section, as shown in Fig. 6.34. The values for the structural parameters refer to the ISOGAI case of NACA64A010 and are reported in Table 6.3. In the present case, writing the equilibrium equations around the elastic center in
6.5. Flutter results

\[ \mu \left( \frac{sb}{U_\infty} \right)^2 M + \frac{1}{V^*} \Omega^2 - \mathbf{E} \left( \frac{sb}{U_\infty} \right) \tilde{\mathbf{q}} = \mathbf{0}, \]  

Equation 6.2 can be recast as a non-linear eigenvalue problem in the non-dimensional variable $V^*$. The problem can be solved in an iterative

---

3$U_\infty$ is the dimensional freestream velocity, $b$ is the semichord, $\omega_\alpha$ is the torsional spring stiffness, $\mu$ is the mass ratio [85, 92], defined as $\mu = m/\pi \rho b^2$.
manner, due to the transcendental dependence of the GAF matrix on the reduced frequency, using the V-g method. Starting point is that at the flutter margin the system is neutrally stable, therefore \( s = j\omega \), where \( j \) is the imaginary unit and \( \omega \) is the (circular) frequency. The equation becomes then:

\[
\left[ -\mu (\omega b / U_\infty)^2 M + \frac{1}{V_{\infty}^2} \Omega^2 - E(j\omega b / U_\infty) \right] \ddot{q} = 0.
\]  

(6.3)
6.5. Flutter results

Figure 6.34: Profile restrained in bending and torsional motion by springs of stiffness $K_h$ and $K_\alpha$, applied at the elastic center, located at distance $ba$ (positive when aft of midchord) from the midchord. Also shown the position of the center of gravity $x_\alpha$ (positive when aft of midchord).

<table>
<thead>
<tr>
<th>Structural parameters for the ISOGAI case of the NACA64A010 profile.</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a = -2.0$, see Fig. 6.34</td>
</tr>
<tr>
<td>$x_\alpha = -0.2$, see Fig. 6.34</td>
</tr>
<tr>
<td>$r_{cg}^2 = 0.24$ dimensionless radius of gyration ratio with respect to the center of gravity.</td>
</tr>
<tr>
<td>$\omega_h/\omega_\alpha = 1.0$ ratio between the uncoupled flexural and torsional frequencies.</td>
</tr>
<tr>
<td>$\omega_h = \sqrt{K_h/m}$, $m$= mass of the profile per unit span.</td>
</tr>
<tr>
<td>$\omega_\alpha = \sqrt{K_\alpha/I_\alpha}$, $I_\alpha$ = moment of inertia around the elastic axis.</td>
</tr>
</tbody>
</table>

Introducing the reduced frequency $k = \omega h/U_\infty$, left multiplying by $[\Omega^2]^{-1}$ and collecting the terms not multiplied by $V^*$, the equation becomes:

$$\left[ \frac{1}{V^*} I - (\Omega^2)^{-1}(\mu k^2 M + E(jk)) \right] \ddot{q} = 0, \quad (6.4)$$

which can be recast as:

$$\left[ \frac{1}{V^*} I - A(k) \right] \ddot{q} = 0. \quad (6.5)$$

The V-g method consists in determining the eigenvalues $\lambda = 1/V^*$ of the matrix $A(k)$ corresponding to different values of the reduced frequency $k$. Starting from a high value of $k$ (corresponding to a low velocity $U_\infty$, according the definition of reduced frequency $k$), the only eigenvalues $\lambda$ physically consistent are those having a zero imaginary part. The sought flutter velocity $U_{fl} = U_\infty$ is defined by the the lowest velocity $V^*$ corresponding to a purely real eigenvalue $\lambda$.

Following the procedure explained above, Table 6.3 reports the flutter speed index $V^* = U_{fl}/(b\omega_\alpha\sqrt{\mu})$, the reduced frequency and the percentage variation in
The flutter analysis has been repeated for the NACA64A010 airfoil, at an angle of attack of 2 degrees, using the GAF matrix components shown in Figs. 6.22-6.25. Table 6.5 reports the flutter speed index, the reduced frequency and the percentage variation in the flutter speed index with respect to the 1% perturbation case. In this case the variations are more evident. However, they are limited to a maximum of 5%, even when the input amplitude is increased by 50% with respect to the reference case, thus proving the outcomes of the linearity checks. The results also show that the flutter speed index variations due to the differences in the components of the GAF matrix are “robust”. Besides, the flutter speed index has a “tendency” to conservativism, as increasing the input amplitude increases the flutter velocity.

In order to validate the results obtained, Fig. 6.35 shows the comparison of the flutter velocity calculated for the case in Table 6.4 with those obtained from other authors. As reported in Fig. 6.35 the results agree quite well with the results obtained by other authors using a CFD code based on the Euler equations. Besides, the same figure shows that the Doublet Lattice method, unable to resolve correctly the shock position and intensity, i.e. the flow’s static non-linearity, is providing the worst results.

<table>
<thead>
<tr>
<th>Impulse Case/Amplitude</th>
<th>Flutter Speed Index $V^*$</th>
<th>Reduced Frequency</th>
<th>Difference with $V^*$ of case (1)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1- 100%</td>
<td>1.154</td>
<td>0.132</td>
<td>0</td>
</tr>
<tr>
<td>2- 500%</td>
<td>1.152</td>
<td>0.132</td>
<td>0.17%</td>
</tr>
</tbody>
</table>

Table 6.4: Flutter speed indexes corresponding to different impulse amplitudes used for the GAF matrix construction for the 0.21 degree angle of attack case, $M = 0.796$.

<table>
<thead>
<tr>
<th>Impulse Case/Amplitude</th>
<th>Flutter Speed Index $V^*$</th>
<th>Reduced Frequency</th>
<th>Difference with $V^*$ of case (1)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1- 1%</td>
<td>1.410</td>
<td>0.136</td>
<td>0</td>
</tr>
<tr>
<td>2- 25%</td>
<td>1.430</td>
<td>0.137</td>
<td>1.41%</td>
</tr>
<tr>
<td>3- 50%</td>
<td>1.468</td>
<td>0.141</td>
<td>4.11%</td>
</tr>
</tbody>
</table>

Table 6.5: Flutter speed indexes corresponding to different impulse amplitudes used for the GAF matrix construction of the 2 degree angle of attack case, $M = 0.796$. 

The results show that the variation in the flutter speed index is negligible, in agreement with the results of the linearity check shown in Figs. 6.16-6.19.
6.6 Conclusions

The flutter results presented for the NACA64A010 airfoil show that for the particular case analyzed the aerodynamic operator can be considered linear with respect to perturbations of the displacements around the steady condition, as long as flow separations or shock boundary layer interactions do not occur, i.e. an Euler model for the flow-field can be assumed. This linear behavior can be attributed to the small excursion of the shock from its steady position, regardless of the small or large displacements given to the airfoil. Indeed the shock motion is limited to less than 2 percent in the worst case analyzed. In this situation a flutter analysis can therefore be performed using the traditional modal approach which leads to the definition of the GAF matrix. In the particular case of weak shocks, the aerodynamic modeling methods like the Field Panel Method, based on a linearized flow model around a steady transonic condition and able to provide directly the GAF matrix, thus avoiding system identification procedures, can be particularly suitable for preliminary design application. Differently, a system identification approach can be used, providing a CFD code with suitable inputs and recording the outputs. A linear system identification tool can be used producing results that have been shown to be robust and conservative.
Chapter 7

Conclusions

In the preliminary phase of the design process, the capability to perform accurate aeroelastic stability analyses at a limited computational cost and with the capability of easily updating the models (structural and aerodynamic), could undoubtedly improve the quality of the design. The occurrence of bad surprises or unexpected outcomes at later stages can be minimized.

In order to enable this capability, an approach has been suggested in the present work.

A KBE approach to the design process, offers a solution for the automation of repetitive operations typical of the preliminary design phase and certainly time consuming, giving the designer the possibility to go beyond the natural boundaries set up by past experiences.

The use of the solid p-elements for the structural analysis, fully supports the KBE design method, reducing the operations needed to transform a geometrical model into a product model.

The flexibility offered by solid p-elements in modeling any kind of structural element such as shell-, beam-, point-like, or full solid elements has reduced the number of options and choices to be made at the meshing stage. Less information needs to be implemented at the product model level and therefore a high level of automation has been achieved. The use of solid p-elements solves the consistency and the convergence problems connected with finite elements analysis, and gives the possibility to create simplified models for stiffened structures.

With respect to these features, the present modeling and analysis approach based on the use of only one type of solid p-formulation finite element can be referred as an Element Independent Finite Element approach.

The possibility offered by the p-elements to run adaptive analyses with error estimation has made it possible to perform the convergence analyses on complex configurations, like a complete wing without the burden of implementing a recursive meshing strategy, limiting the number of steps during the evaluation process of a certain configuration.

The robustness and flexibility demonstrated by the solid p-elements in computing an accurate solution, using very coarse meshes, with highly distorted and very high aspect ratio finite elements, gave the possibility to avoid the “stan-
standard" meshing process, as it was just sufficient to generate a solid model where all the connections between the different solid patches were found and directly translated it into a finite element model.

With respect to the aerodynamic modeling, in the transonic range the use of CFD codes can become a modeling necessity, representing a huge factor of slowing down the design analysis and the properties evaluation process. When the shock motions are small and no shock boundary-layer interactions are present, the CFD analysis can be coupled with a linear system identification procedure, which has shown to be accurate enough to capture the dynamics of a profile around a transonic steady condition.

A traditional flutter analysis can in this case be performed deriving the GAF matrix. In the special case of weak shocks and small shock motions the system identification procedures can be avoided as field panel methods can directly provide the GAF matrix to perform flutter analysis.
Chapter 8

Recommendations

Although the behavior of the solid p-elements has been extensively investigated, the job is not finished yet. An evaluation for a complete aircraft is still missing, and this is connected to the possibility to make use of hybrid meshes.

Besides, the error estimator that is used in Nastran is based on the convergence of the energy. This has shown to be too strict when modal shapes and frequencies must be captured. The criterion can be “relaxed” as the frequencies converge much faster than the energy norms, thus a more suitable convergence criterion can be formulated.

Moreover, the work in the present thesis has been performed using a commercial package. The use of an off-the-shelf package has given the possibility to dedicate the time needed for the development of such a code in doing more investigations about the behavior of the solid p-elements. For further steps, it would be good, especially for research purposes, to develop an in house code, for which there is access to all the source files. Having access to these files would give the possibility to focus for example on the numerical routines used to compute the solutions. Indeed the most asked question about the use of p-elements is about the computational time. Doubling the degrees of freedom the computational time increases following a non-linear law. The research on the numerical algorithms for solving linear systems is an ongoing world, and having the possibility to replace the sparse decomposition solver used within Nastran with a more efficient one would make the solid p-elements more attractive than they are now.

Regarding the ICAD software used to create the solid-model generator, it must be remarked that the decision to use surfaces instead of solids, was due to the inability of the software to handle solids in a flexible manner and this made things easier on one side, as surfaces are simpler objects to handle, and more difficult on the other, as it forced to do a lot more additional operations. For example finding the intersections between the different solid patches, which eventually is the most important operation to be performed (and also the only one whenever the “standard” meshing process can be avoided), are much better done when real solids could be efficiently handled. It would be therefore better to look at other KBE systems available on the market, for which these operations
could be performed in an easy way, also because the ICAD system is not anymore available on the market.

From the CFD side, such codes are notoriously slower than a structural solver, and their use is not the most suitable in a preliminary design environment. In order to reduce the computational time of an aeroelastic solution, it is useful to make use of system identification techniques. For the purposes of the present work, it has been used the simplest procedure (transfer function determination) based on a Single-Input, Single-Output approach, but it would be advisable to make use of a Multi-Mode excitation technique where more modes are excited simultaneously in the CFD code and the contribution of the single modes can be separated in the output. This technique is already available in the System Identification branch. Moreover, as in some case the system identification can be completely avoided without loosing anything in the description of the phenomena, it would be advisable to develop a procedure that is able to estimate whether the shock position is stable or not, or in other words, if the aerodynamic non-linearities are important or not.

Also for the aerodynamic part, a commercial code has been used. Again, on one hand it was possible to transfer the code development times into more investigation time, and on the other hand the code could be used as it is with its limits for example in the accuracy of the numerical schemes (the time integration was only first order accurate), flexibility in the pre- and post-processing operations.

An efficient solution to the need of standard software (a finite element code having p-elements for example, or a CFD code based on the Euler equations), which gives access to all the source files and that can be modified for the needs of the research subject carried out, is the one offered by some universities and research centers. Some institutes indeed developed codes which are made available for research purposes, to all academic institutions and also industries who are interested.
Appendix A

System Identification Tool for a DEE

The System Identification cycle is made of different steps. In this appendix we want to put the attention on three important aspects of the cycle, that are the

- Definition of the input function needed to excite the system.
- Definition of the parameters needed to set-up the system identification procedure.
- The mesh generation

A.1 Definition of the input function.

For systems identification the CFD code is provided with an input function of the type:

$$u(x,t) = \sum_{n=1}^{\kappa} \Psi_n(x) q_n(t)$$  (A.1)

where $\Psi(x)$ is the $n$-th mode determined through a modal analysis, and $q(t)$ is the time-law chosen. As stated before the time law is defined by impulse-like function. Different choices are available for it. For example in the papers of Targusi [93] and Silva [94] the following laws are chosen:

$$q(t) = Amp \cdot \left[ t \frac{T_{imp}}{2} \right]^2 \cdot e^{\left[ 2\pi - \frac{t}{T_{imp}} \right]}$$  (A.2)

$$q(t) = \frac{Amp}{2} \cdot \left[ 1 - \cos \left( 2\pi \frac{t}{T_{imp}} \right) \right]$$  (A.3)

$$q(t) = Amp \cdot e^{\left| w(t - T_0)^2 \right|}$$  (A.4)

where $T_{imp}$ is the time interval during which the function is not zero, $Amp$ is the amplitude of the input function, $T_0 = T_{imp}/2$ is the time at which the
Figure A.1: Comparison of the impulse–like signals \( A.2 \) (referred as Impulse-1), \( A.3 \) (referred as 1-cos), with \( T_{imp} = 0.08 \), \( Amp = 1 \), \( K = 0.16 \), and their first and second derivatives.

Apart from the signal defined in Eq. A.4 for which no information on the parameter \( w \) is given, it has to be said that although the other two input signals, defined in Eq. A.2 (referred as impulse-1) and defined in Eq. A.3 (referred as 1-cos), have the same shape, as shown in Fig. A.1a, they have very different effects on the response of the aerodynamic system. Indeed Fig. A.1c shows a discontinuity of the second derivative for the 1-cos signal at time \( t = T_{imp} \), whereas both signals have a non zero value of the second derivative at time \( t = 0 \).

As the generalized forces depend also on the first and second time derivatives of the modal displacement input, particular attention should be put to these derivatives especially at the time zero and the time \( T_{imp} \), where the impulse is connected to the zero–part of the input signal. At these time instants it must be assured that at least the first and second derivatives are continuous otherwise, although the (impulse–like) input has been tailored to have a certain frequency content (with the physical time–step and minimum cell size of the mesh chosen to capture the frequency content of the input function), the response of the system will have a higher frequency content and will lead to a failure of the identification process unless time-step and cell size are conveniently reduced. In
A.1. Definition of the input function.

To handle this issue in a very simple and straightforward manner, in the present work the input function has been expressed as:

$$ q(t) = \text{Amp} \cdot e^{\left( -t - \frac{T_{imp}}{K} \right)^2} $$

(A.5)

Being $K$ ($0 < K < 1$) a factor that determines the smoothness of any derivative order of the input signal at the times $t = 0$ and $t = T_{imp}$. Choosing a suitable value for the factor $K$, all the derivatives can be set to a (numerical) zero value at the times $t = 0$ and $t = T_{imp}$. As an example Fig. A.2 shows the comparison of the signal $A.5$ with the derivatives of the signals $A.2$ and $A.3$. As can be seen the signal $A.5$ (referred as Impulse-2) eliminates all the discontinuities at times $t = 0$ and $t = T_{imp}$, thus avoiding the excitation of unnecessary and unwanted high frequencies in the system to be identified.

---

1This law has been derived from a function of the same type, used for verification purposes in [93].
A.2 Parameters needed for the system identification procedure.

The parameters to be controlled during the system identification cycle are reported in the following table:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T_{imp}$</td>
<td>Time interval in which the input signal is not zero</td>
</tr>
<tr>
<td>$N_{camp}$</td>
<td>Number of time samples</td>
</tr>
<tr>
<td>$N_{imp}$</td>
<td>Number of samples inside $T_{imp}$</td>
</tr>
<tr>
<td>$\Delta t = T_{imp}/N_{imp}$</td>
<td>Time step for discretization</td>
</tr>
<tr>
<td>$T_{obs} = \Delta t \cdot N_{camp}$</td>
<td>Time interval in which the phenomenon is recorded</td>
</tr>
<tr>
<td>$\Delta f = 1/T_{obs}$</td>
<td>Frequency resolution</td>
</tr>
</tbody>
</table>

- $T_{imp}$: time interval in which the input function is not zero. It is chosen such that the frequency content of the input function covers the frequency range defined by the highest frequency mode of interest;

- $\Delta t$: time step. A physical time-step and a numerical time step can be defined. The physical time step is the one calculated in order to correctly capture the frequency content of the response of the system. In the linear case it is defined by the maximum frequency of interest. In the non-linear case it is half of the time step of the linear case. As it will be shown later its value is important for the creation of the mesh. The numerical time-step is defined as the physical time step reduced at least by a factor ten as Fluent requires that the time step is one-tenth of the smallest time constant of the system, defined in our case by the maximum frequency of interest. This value is considered just for starting sensitivity analyses on the time step choice;

- $T_{obs}$: observation time. It is determined by different requests. It should be chosen to comprise the minimum frequency of interest, i.e. the frequency of the lowest mode of interest. Besides it should be long enough to have both the response of the system vanishing completely, and an acceptable frequency resolution for the response of the system;

- $N_{camp}$: number of samples in time domain. It is determined dividing the observation time $T_{obs}$ by the time step $\Delta t$.

A.3 Mesh generation for a system identification tool

A possible methodology for the generation of the mesh around a profile, for the purposes of system identification, has been implemented as described below.
A.3. Mesh generation for a system identification tool

Parameters defining the computational domain.

Multiblock definition of the computational domain.

Figure A.3: Multiblock definition of the computational domain and parameters needed to define it.

First step is the creation of a control volume (i.e. a surface with zero thickness) around the profile. Shapes commonly used are the circular shape, due to the easiness in its creation, or a parabola shaped area which allows more control on its size, according to the parameters described hereafter. The size of parabola shaped control area is governed by the three parameters, shown in Fig. A.3a:

- The distance between the leading edge and the upstream vertex of the parabola, measured in multiples of the chord ($N_{cl}$ parameter in Fig. A.3a);
- The downstream distance measured from the trailing edge in multiples of the chord, i.e. the size of the wake ($W_{len}$ parameter in Fig. A.3a);
- The upwards distance measured from the profile, in multiples of the chord ($N_{cu}$ parameter in Fig. A.3a).

The choice of these three parameters can be done such that the disturbance in the pressure field generated by the profile doesn’t hit the border of the domain, with the wake long enough to capture the steady effects. This can be done in two different ways: the first is choosing “arbitrarily” big values for these parameters but this is not an efficient way as this would mean high computational time; the second is based on some sensitivity analysis on the value of the parameters, and it could be done running a steady analysis using a little number of iterations and a rather coarse mesh. Clearly this is a more efficient way as with a small computational cost oversizing of the mesh and consequently waste of time can be avoided. Once the area has been created it is divided into four parts (multi-blocks), Fig. A.3b in order to have a better control on the mesh that is to be created.

For all the computations performed it was chosen to use a structured mesh. It has the advantage to be more controllable than an unstructured one although the control of the mesh requires the use of many parameters. The four surfaces created are four-sided and so they can be easily meshed. The areas are meshed specifying the number of elements on each of their edges. This is done using information such as Mach number of interest, frequency range of the modes of
interest and other parameters necessary to perform the identification. A Matlab routine has been built to calculate these parameters, and it is explained in the following. In the previous section it has been described how to determine the time step to use. Its choice influences directly the size of the mesh. Indeed in the unsteady case the signal is shed along the wake with a frequency that depends on the oscillation frequency of the airfoil. So the largest element of the wake has been sized to capture this signal. If \( U_\infty \) is the freestream speed far ahead of the airfoil and \( \nu \) is the oscillation frequency (in Hz) of the airfoil, the largest element of the wake must respect the condition

\[
\Delta x \leq \frac{U_\infty}{\nu}
\]  

being \( \nu = 2/\Delta t \) (\( \Delta t \) = time step). Once the largest element size has been defined, it is possible to determine the number of elements along the wake length, knowing the law with which the elements are distributed along the wake length. Gambit offers several options regarding the law used for the distribution of the elements along a generic edge. One of these laws, consists in specifying the ratio between two consecutive elements (named successive ratio) and find the number of elements on the edge considering that:

\[
L = \Delta x_0 + \Delta x_1 + \cdots + \Delta x_n = \sum_{i=0}^{n} \Delta x_i
\]

where \( L \) is the edge’s length and \( \Delta x_i \) is the generic element’s length. Once assigned the constant ratio between two consecutive elements \( i \) and \( i + 1 \), it results that:

\[
\frac{\Delta x_i}{\Delta x_{i+1}} = K \quad \text{and} \quad K \leq 1
\]

and therefore:

\[
\Delta x_0 = K^p \Delta x_p, \quad p = 1 \cdots n
\]

Substituting inside the sum and remembering that the partial sum of a geometric series is:

\[
S_n = \sum_{i=0}^{n} K^i = \frac{1 - K^{n+1}}{1 - K}
\]

the number of elements on the edge considered is:

\[
N_{el} = \frac{\ln \left( 1 - \frac{(1-K)^{n+1} L}{\Delta x_n} \right)}{\ln K}
\]

being \( \Delta x_n \leq \frac{U_\infty}{\nu} \). Once determined the number of elements and the successive ratio, the size of the first element of the wake, attached to the trailing edge, is known.
The element on the wake closest to the trailing edge determines the size of the element of the profile adjacent to the trailing edge, Fig. A.4. Again, fixed the stretching function on the profile and given the half profile length, it is possible to determine the number of elements to be placed on it. In our application it has been chosen a successive ratio law, which fix the ratio of the size of two successive elements.

The number of elements on the dashed lines in Fig. A.3b is the same and it is determined by the particular model for the fluid, viscid or inviscid, that has to be used. In both cases a stretching function is needed to concentrate the cells in the regions adjacent to the profile and the wake. In the viscous case this requirement is more strict as the solver requires that at least a few points fall inside the boundary layer whose thickness must be estimated first and verified during the run. In the inviscid case this requirement can be relaxed taking care that the cell adjacent to the leading edge have a not too high aspect ratio as this can slow down the local convergence of the solution in a region of high pressure gradients. In this way, as long as the parameters for the stretching functions are fixed, the mesh around the profile can be controlled just using one parameter that is the stretching ratio used on the wake. Varying the stretching ratio different meshes can be generated and a sensitivity analysis on the number of cells can be performed, just using one parameter.

All the information necessary for the mesh creation, number of elements and stretching functions the edges are handled by the Matlab routine which generates a journal file containing all the instructions ready to be executed by the mesh generator, in the present case the Gambit software, the mesh generator for the Fluent CFD software that has been used to perform the CFD computations presented in this work. Running Gambit in batch, a mesh is generated. The only interaction with the user is for checking the mesh. Besides a Fluent script can be generated containing information about the kind of analysis, steady or unsteady, solver employed, number of iterations and so on. For the steady solution an implicit coupled solver is selected with a second order upwind numerical
scheme. For now the analysis is limited to the inviscid case. The Courant number is depending on the flow regime: increasing the Mach number the maximum Courant number usable decreases. In the unsteady case, time step determined through the Matlab procedure is added in the batch script, together with the number of sub-iterations per time step and the total number of time steps. The mesh generation process becomes in the end completely parametric and flexible.
Bibliography


Summary

Elements of Automated 
Aeroelastic Analysis 
in Aircraft Preliminary Design

by Paolo Lisandrin

The possibility to perform automated computations is an important aid in aircraft preliminary design phase. Limited experimental data are available in this phase and performing computations with high accuracy and limited computational costs is crucial at this stage. This is particularly important for the aeroelastic stability analysis in the transonic regime, due to the multidisciplinary nature of the aeroelastic analysis and the complexity of the transonic flow regime where shock waves appear.

An aeroelastic stability analysis tool can be thought as made of a structural analysis block and an aerodynamic analysis block. In the present thesis the structural analysis block is a Finite Element (FE) analysis, whereas the Aerodynamic analysis block is made of a Computational Fluid Dynamics (CFD) analysis. To avoid the direct coupling of a CFD code and a FE code, not affordable in a preliminary design environment, each block is replaced by a Reduced Order Model (ROM). For the structural block the ROM consists of a modal model derived from the FE modal analysis.

A new approach is proposed to achieve a high level of automation for the modal analysis of a structure. It consists in using just one type of solid, i.e. brick, p-formulation, finite element. The three-dimensional geometry of this type of element guarantees flexibility in the element type representation (thick or thin shells, beams, point-like elements), whereas the p-formulation approach enables the use of an adaptive analysis method. Convergence of the FE solution is achieved through the increase of the polynomial order of the shape functions while keeping fixed the geometry of the finite element model. Convergence checks become affordable even for complex structures like a complete aircraft because no mesh re-generation is required.

P-formulation finite elements are available in the commercial FE package MSC-NASTRAN. Using this software, the proposed FE analysis approach is tested on the modal analysis of isolated panels, made of an isotropic material and having different types of geometries. The panels examined are both thin
and thick walled, having zero, single and double curvature. The results of the modal analysis are validated against the results obtained using models built using linear shell finite elements. For the shell models convergence is achieved increasing the number of elements, thus regenerating the mesh for each computation (h-convergence). The solid p-formulation models show that convergence is achieved using only one very coarse mesh with few adaptive steps, whereas for the shell models convergence is achieved requiring more steps, with the need of regenerating a mesh at each computation.

New generation aircrafts have an increasing percentage of their airframe made of composite materials. The modal analysis of a laminate using brick elements requires the description of the material properties along the laminate’s thickness. To avoid the modeling of each lamina of a laminate as a solid, an equivalencing process is formulated to match the membrane behaviour of the equivalent model with that of the laminate’s full model. The equivalent laminate is represented by a one layer laminate made of an orthotropic material. The equivalencing process is validated on the computation of the modal frequencies of a sandwich plate with composite facings. Two different FE models of the sandwich plate are generated using solid p-formulation finite elements and linear shell elements. The results show that the equivalent model gives the same result of a full model and again the solid model converges much faster than the shell model.

Aeronautical structures make use of stiffeners to prevent buckling of thin-walled panels. Modeling stiffeners using solid elements would generate a much too complex model with a level of accuracy not needed in a modal analysis. Therefore an equivalencing method to create dynamically equivalent models of stiffened panels in wing-boxes is developed. The equivalent model is obtained smearing the stiffeners along the entire surface of the stiffened panel and determining the geometrical properties (thickness) and the elastic properties of the equivalent layer by equivalencing the structural axial, bending, torsional stiffnesses and the total mass of the full model. The equivalencing process is tested over a squared wing-box having a stiffened skin. A modal analysis performed over the full model and over the wing-box with the equivalent model shows that the results differs from each other by few percent, thus within engineering acceptance.

The FE solid modeling and analysis method is tested on wing-like structures. In order to be able to generate solid geometries and the relevant finite element model, a dedicated solid mesh generator is created using the Knowledge Based Engineering (KBE) software ICAD. The solid mesh generator accepts as an input a surface model, either as a set of IGES files coming from another CAD package, or from ICAD itself using a Multi-Model Generator already developed in ICAD. Using the CAD capabilities and the KBE features of ICAD, the solid mesh generator creates a mesh of the wing considered and then a FORTRAN routine translates the mesh into a MSC-NASTRAN input file, ready to be executed. Different wing geometries are generated and the results of the modal analysis are compared with the one performed using linear shell elements. The results show that the solid p-elements converge much faster than the shell models. In a test done over the modal analysis of a Blended Wing Body outer wing, the
convergence rate slows down, becoming comparable with the one of the shell elements. This happens because very low local stiffness, unacceptable from an engineering point of view, are present in the model. Nevertheless the convergence speed is not the fundamental issue in automation.

The use of CFD for the aeroelastic stability analysis offers the possibility to analyse any kind of shock strength and shock motion at the price of a computational cost that is higher than commonly used analysis methods, like the Full Potential equations for example. Even if parallel, unstructured formulations are available on the market, still the CPU times can be prohibitive. ROMs can be derived using system identification techniques. The ROM is intended to describe the dynamics of an aerodynamic system around a steady configuration. The ROM can be derived using either linear or non-linear system identification methods. It is important to draw a boundary between the linear and non-linear identification techniques. Indeed the use of a linear system identification method translates in a remarkable saving of computational time. Therefore an investigation is performed over the importance of the non-linear terms of the flow equations by using simple Fourier transforms. Profile motion, in pitch and plunge modes, is studied using different excitation inputs. Inputs considered are a sinusoidal oscillation in pitch and plunge modes and a Gaussian shaped function, simulating an impulse tailored to have prescribed frequency content. Linearity checks are performed on the response to the sinusoidal input. Using the enlarged impulse shaped signal, the sensitivity of the Generalized Aerodynamic Forces (GAF) matrix is checked using different amplitude signals. The same sensitivity is performed on the results of the flutter analysis using the different GAF matrices obtained. The flutter velocities computed using the different GAF matrices are almost independent of the input amplitude, showing that the non-linear effects are not important for this case.
Samenvatting

Aspecten van Geautomatiseerde
Aeroelastische Analyse
in de Vliegtuig-Voorontwerp fase

door Paolo Lisandrin

De mogelijkheid om geautomatiseerd berekeningen te doen is een belangrijk hulpmiddel in de vliegtuigvoorziening-fase. In dit stadium is er slechts beperkte informatie beschikbaar en het uitvoeren van analyses met een hoge nauwkeurigheid maar met beperkte kosten is essentieel. Dit is vooral belangrijk voor de analyse van de aero-elastische stabiliteit in het transsonic gebied, vanwege het multi-disciplinaire character van de aeroelastische analyse en de complexiteit van het transsonic stromingsgebied waarin schokgolven optreden.

Een aeroelastische stabiliteitsanalyse tool bestaat uit een structurele analyse deel en een aerodynamische analyse deel. In deze dissertation is het structurele analyse deel ingevuld met een eindige-elementen (FE) analyse en het aerodynamische analyse deel met een Computational Fluid Dynamics (CFD) analyse. Om de noodzaak van een directe koppeling tussen CFD en FE code te voorkomen is elk van beide analyse delen vervangen door een Reduced Order Model (ROM).

Om de noodzaak van een directe koppeling tussen CFD en FE code te voorkomen is elk van beide analyse delen vervangen door een Reduced Order Model (ROM).

Voor het structurele analyse deel is het ROM afgeleid van een eindige-elementen modaalanalyse.

Een nieuwe aanpak is voorgesteld voor het verkrijgen van een hoge mate van automatisering van de modaalanalyse van een constructie. Gebruik wordt gemaakt van slechts n type volumelement gebaseerd op de p-formulering. De drie-dimensionale geometrie van dit type element garandeert flexibiliteit in het bereik van de te modelleren constructieve elementen (dikwandige en dunwandige schalen, balken en puntachtige elementen), waarbij de p-formulering het gebruik van adaptieve analyses mogelijk maakt. Convergentie van de FE-oplossing wordt verkregen door het ophogen van de orde van de shape functies zonder de geometrie van de afzonderlijke elementen aan te passen. Convergente analyse wordt op deze manier bereikbaar voor complexe constructies zoals vliegtuigrompen en v-vleugels omdat het meerdere malen discretiseren van de constructie wordt voorkomen.

Eindige elementen op basis van de p-formulering zijn beschikbaar in het commerciële eindige-elementen pakket MSC-NASTRAN. De voorgestelde aan-
pak is geverifieerd aan de hand van een modaal analyse op enkele panelen van verschillende geometrie en bestaand uit isotroop materiaal. Zowel dikke als dunwandige, niet, enkel en dubbelgekromde panelen zijn onderzocht. De resultaten zijn vergeleken met die verkregen met modellen opgebouwd uit lineaire schaal elementen. Voor de schaalelementen convergentie is verkregen door het steps-gewijs vergroten van het aantal elementen, dus het opnieuw discretiseren van de constructie. Dit is de zogeheten h-convergentie. De resultaten met de p-elementen laten zien dat convergentie verkregen wordt gebruikmakend van een enkel, grog grid met een beperkt aantal adaptieve stappen (vergroting van de orde van de shape-functies polynomen). Convergentie voor de schaalelementen vereist meer stappen met de noodzaak voor iedere stap de constructie opnieuw te discretiseren.

Nieuwe generaties vliegtuigen hebben een steeds groter percentage van hun constructie opgebouwd uit composites. Voor de modaalanalyse van een laminaat met gebruikmaking van volumeelementen is het noodzakelijk het verloop van de materiaal-eigenschappen over de dikte van de elementen te beschrijven. Om het modelleren van elke laag van een composietconstructie als aparte laag volumeelementen te vermijden, is een equivalencing methode opgesteld om het membraangedrag van model en constructie te laten overeenkomen gebruikmakend van slechts n laag volumeelementen opgebouwd uit orthotroop materiaal. Het equivalencing process is gevalideerd met modaalanalyses van sandwichpanelen met composiet-huiden. Er zijn twee verschillende FE-modellen van de sandwich plate gemaakt gebruikmakend van de op p-formulering gebaseerde elementen en lineaire schaalelementen. De resultaten laten zien dat het equivalent model dezelfde resultaten oplevert als het volledige model en ook hier het op volumeelementen gebaseerde model sneller convergeert dan het op schalen gebaseerde model.

In luchtvaartconstructies wordt gebruik gemaakt van verstijvers om het knikken van dunwandige platen te voorkomen. Het modelleren van dergelijke verstijvers met volumeelementen levert een te complex model op met een nauwkeurigheidsniveau dat onnodig hoog is voor een modaal analyse. Hierom is een equivalencing methode ontwikkeld waarmee dynamisch equivalent model wordt verkregen door de verstijvers uit te smeren over het gehele oppervlak van het verstijfde paneel. De geometrische eigenschappen (dieplaatdikte) en de elastische eigenschappen van het equivalent model worden verkregen door het gelijkstellen van de constructieve axiale, buig- en torsiestijfheid en de totale massa van equivalent model en volledig model. De methode is getest op een vierkante vleugel van 2m lang met een verstijfd vleugel. Een modaal analyse, gedaan op het equivalent model, laat zien dat de resultaten slechts enkele procenten verschillen en dus in een engineering omgeving toepasbaar zijn.

De methode gebaseerd op volumeelementen is ook getest op vleugelachterste constructies. Ten einde geometrische volume modellen (solid models) en de gerelateerde volumeelementen te kunnen genereren, is een speciale mesh-generator voor volumeelementen gebouwd met behulp van het Knowledge Based Engineering softwarepakket ICAD. De mesh generator accepteert oppervlaktemodellen
als invoer. Deze oppervlaktemodellen kunnen als een verzameling IGES files vanuit een ander software pakket worden aangeleverd of vanuit reeds bestaande zogeheten Multi-Model Generators die door de leerstoel zijn ontwikkeld. Gebruikmakend van de CAD en KBE features in ICAD, maakt de solid mesh generator een op volumeelementen gebaseerde discretisatie van de vleugel die vervolgens door een FORTRAN routine wordt omgezet in een invoerbestand voor MSC-NASTRAN. Op deze manier zijn modellen van verschillende vleugels gemaakt en de resultaten van de modaalanalyse zijn vergeleken met die verkregen met op lineaire schaalelementen gebaseerde modellen. De resultaten laten zien dat volumeelementen gebaseerd op de p-formulering veel sneller convergeren dan die gebaseerd op lineaire schaalelementen. In een test op een complexe structuur zoals een Blended Wing Body buitenvleugel blijkt wel dat de convergentiesnelheid afneemt en vergelijkbaar wordt met die van lineaire schaalelementen. Dit effect treedt op doordat er lokaal te lage stijfheden in het model voorkomen die vanuit engineering standpunt ook niet acceptabel zijn. Overigens is convergentiesnelheid niet een fundamentele eigenschap in automatisering.

Het gebruik van CFD in de analyse van de aeroelastische stabiliteit biedt de mogelijkheid elke schoksterkte en schokbeweging te onderzoeken maar met gerelateerde rekenkosten die aanzienlijk hoger liggen dan voor de normaal gebruikte methoden zoals potentiastromingsberekeningen. Hoewel parallele en ongestructureerde formuleringen beschikbaar zijn op de markt, kunnen de reken- tijden toepassing in de weg staan. ROMs kunnen afgeleid worden met behulp van systeemidentificatie methoden. De ROM is bedoeld om de dynamica van een aerodynamisch systeem rond een stabiele situatie te beschrijven. De ROM kan worden afgeleid met lineaire en niet-lineaire identificatie technieken. Het gebruik van lineaire systeemidentificatie methode vertaalt zich in een aanzienlijke besparing van rekenkosten. Om deze reden is onderzoek gedaan naar de importante van de niet-lineaire termen in de stromingsvergelijkingen met eenvoudige Fourier transformaties. Bewegingen van een profiel in pitch en plunge mode zijn bestudeerd aan de hand van verschillende excitaties. Onder de bestudeerde invoor zijn een sinusvormige trailling in pitch en plunge mode en een Gauss-achtige functie waarmee een impuls met op maat voorgeschreven frequentieinhoud kan worden gesimuleerd. Controles op lineariteit zijn uitgevoerd aan de hand van de sinusvormige excitatie, gebruikmakend van het uitgesmeerde impulsformige signaal. De gevoeligheid van de generaliseerde aerodynamische krachten matrix is bestudeerd gebruikmakend van signalen met verschillende amplitudes. Een soortgelijke gevoeligheid analyse is gedaan op basis van de resultaten van flutter analyses waarin gebruik gemaakt is van de eerder genoemde matrices. De resultaten laten zien dat ze zo goed als volledig onafhankelijk zijn van de amplitude van de invoer wat aantoont dat de niet-lineaire effecten niet ter zake doende zijn in dit geval.
Publications


P. Lisandrin and M. van Tooren, High-order finite elements for structural dynamics applications, *to appear on the Journal of Aircraft*. 