Composite Structures 92 (2010) 2016–2026

Contents lists available at ScienceDirect

Composite Structures

journal homepage: www.elsevier.com/locate/compstruct

Numerical simulation of bird strike damage prediction in airplane flap structure

I. Smojver *, D. Ivančević

Faculty of Mechanical Engineering and Naval Architecture, University of Zagreb, I. Lučića 5, HR-10000 Zagreb, Croatia

ARTICLE INFO

Article history:
Available online 11 December 2009

Keywords:
Airplane structures
Bird strike damage
Sandwich structure
Composite structure
Explicit integration

ABSTRACT

Numerical analyses of bird impact damage in complex aircraft structures have been performed using ABAQUS/Explicit. A Lagrangian formulation was used for the bird model in combination with various material models. Several failure and damage modes have been considered for different material models used in the inboard flap of a typical large transport aircraft. A submodeling approach has been used to reduce computational time. Parametric analyses have been performed using different bird sizes, impact locations and velocity vectors.

1. Introduction

Bird strike events present a possible source of risk to the safety of air transport. Although most bird strike events involve relatively small birds, thus not causing catastrophic consequences, the severity of damage generated by collisions with larger birds can not be neglected. In order to ensure tolerance to bird strike damage, aircraft structures have to fulfil the airworthiness specifications prescribed by FAA or JAA. According to FAR 25, Sub-part 25.571, flap structures of large transport aircraft have to withstand an impact with a 4 lb (1.81 kg) bird at normal operating speeds [1]. Most bird strike incidents occur during the takeoff and landing phases of the flight in which the flaps are deployed and exposed to bird impacts.

This paper deals with the numerical prediction of structural behaviour and damage caused by bird strikes in a typical large airliner flap structure. Numerical bird strike simulations are gaining increased importance especially during the certification phase of new aircraft design, where finite element modelling of bird strike damage could reduce, or even replace, time consuming and costly gas gun experiments on prototype components of a new aircraft. Nonlinear transient analyses have been performed using ABAQUS/Explicit.

Finite element bird strike simulations present an interaction of several complex numerical problems including contact, various damage initiation and evolution models, finite element failure and removal of failed elements, impact on sandwich structures etc.

Correct modelling of fluid-like bird behaviour during the impact is essential to accurately predict the damage caused by bird strikes. There are three main approaches to numerically simulate the bird in an impact event: the Lagrangian approach, the Arbitrary Lagrangian Eulerian (ALE) approach and the Smooth Particle Hydrodynamics (SPH) method. The first approach uses Lagrangian finite element formulation for the bird. The main disadvantage of this method is the large deformation of bird finite element mesh during the impact. If the mesh distortion is too large, excessively distorted elements will decrease the stable time increment of explicit integration algorithms to an unacceptable level or even lead to numerical problems and premature analysis termination. To prevent such numerical instabilities, an automatic trial and error procedure has been implemented in [2] to remove excessively distorted elements from the mesh.

In the ALE approach the bird material flows relative to an Eulerian mesh, thereby avoiding large mesh distortion. The impacting loads are transferred to the Lagrangian mesh of the impacted structure through an ALE coupling algorithm. This bird strike modelling approach has been used in e.g. [3–5].

The SPH, a more recent approach to the bird strike modelling problem, is a mesh free method based on the Lagrangian formulation in which the finite elements have been replaced by a set of discrete, mutually interacting particles. Due to the fact that this approach is a grid-less method, it is well suited for impact problems where occurrence of large distortions is expected. This method has been used for example in [1,6]. The main drawback of ALE and SPH compared to the pure Lagrangian description of the bird behaviour is the increased computational time of such simulations, although providing more numerical stability.

The work in this paper followed the Lagrangian bird modelling approach. Numerical difficulties associated with excessive element distortion have been bypassed by using viscous hourglass and distortion control.

The numerical procedure simulating bird strikes has been applied on a very detailed model consisting of composite laminates, sandwich structures and metallic structural items, thus presenting

* Corresponding author. Tel./fax: +385 1 6168 267.
E-mail address: ismojver@fsb.hr (I. Smojver).

0263-8223/$ - see front matter © 2009 Elsevier Ltd. All rights reserved.
doi:10.1016/j.compstruct.2009.12.006

a real aeronautical component consisting of multiple materials. Various failure and damage modes for different material models have been considered. It has not been possible to perform the experimental verification of the numerical damage model on the actual airplane component, as it would have been extremely expensive. Therefore, the numerical procedure has been verified partially through comparison with available references. Although this type of verification does not prove the viability of the procedure as a whole, it is the good indicator of the modelling accuracy of its particular components.

As the bird strike induced damage is localized to the area in which the impact takes place, analyses have been performed on the complete flap model and on substructure models, thereby reducing computational time and output file sizes. In order to reduce the complete finite element model, an automated procedure has been applied which modifies the original ABAQUS input file and produces a new input file consisting of elements in the vicinity of the impact zone. Impacts have been applied on several locations on the flap structure with different bird weights and initial velocity vectors to simulate possible impact scenarios during takeoffs and landings.

2. Numerical model

2.1. Flap finite element model

Flaps of modern large airplanes are usually designed with a great percentage of non-metallic structural elements in order to decrease their weight. The analysed finite element model represents a typical large airliner inboard flap structure. The skins are made of CFRP with variable number of unidirectional layers resulting in gradually variable skin thickness. The upper and lower skins are additionally strengthened with 10 CFRP stringers, of which six are positioned along the upper skin and four are positioned on the lower flap skin. Leading and trailing edges of the flap are designed as sandwich structures. The thin face layers of the leading edge sandwich are also made of CFRP, while Al 2024 alloy was used for the flap trailing edge sandwich face layers. Nomex honeycomb core is used in both sandwich structures. The flap interior structure consists of spars and ribs made of various aluminium alloys. Front and rear spar, as well as ribs which comprise structural items used to connect the flap to the wing structure and hydraulic actuators, are made of Al 7075 alloy with higher mechanical properties, while the remaining ribs and auxiliary spar are made of Al 2024 alloy.

Most of the flap geometry is suitable for shell finite element discretisation. Nevertheless, flap leading and trailing edges are, due to the relatively large thickness of sandwich structures, modelled with three dimensional finite elements. The thin face layers are meshed with three dimensional continuum shell elements which have kinematic and constitutive behaviour similar to the conventional shell elements. These elements enable capturing three dimensional geometry, thus improving accuracy in contact problems. Due to the fact that both continuum shell and solid elements have only translational degrees of freedom, coupling at the interface of face layer and core elements is achieved by sharing the same nodes and thereby eliminating the need for kinematic constraints [7]. The main disadvantage of continuum shell elements is the small element length in the thickness direction, which leads to a decrease in stable time increment for explicit analyses, thereby increasing the computational time. In order to correctly transfer rotational degrees of freedom at the interface of three dimensional and conventional shell elements, an appropriate coupling constraint needs to be provided at their common boundary. These kinematic coupling constraints have been achieved by the tie surface based constraints which are imposed to nodes at the boundary of continuum and conventional shell elements. Fig. 1 shows the flap geometry and three dimensional finite element meshes. The mesh of interior structure is shown in Fig. 2. The total number of elements for the entire model is 109585, of which 69696 are conventional shell, 16020 continuum shell, and 23665 solid elements. Front spar reinforcements are modelled with 204 beam elements.

2.2. Substructure modelling approach

Nonlinear transient analyses on large and complex models demand enormous computation time despite the usage of most modern computers. In order to reduce computational cost and output file sizes, bird strike damage prediction analyses could be accurately performed on smaller parts of the flap structure. This assumption is based on the fact that impact induced damage is localized to the area in the vicinity of the impact location. For the purpose of model reduction, the C# subroutine has been developed which searches through the original ABAQUS input file and removes parts of the finite element mesh which are further away from the impact location. The removed area is user defined and measured from the impact location what enables the creation of arbitrary model sizes. The program combines bird and flap input files into one single input file, which is then submitted through the ABAQUS solver. After selection of desired bird model, initial conditions have to be defined by adjusting the bird model orientation and offset distance from the impacted node. Boundary conditions are assigned to nodes at the newly created boundaries or, if they are appropriate, automatically copied from the original model. This procedure enables a user friendly generation of input files, hence eliminating the usage of the ABAQUS pre-processor and the time demanding procedure of manual creation of smaller models. Fig. 3, left-hand image, shows the simplified block diagram of the substructure generation program. The right-hand image depicts the substructure model used for impact analyses in this particular case, where the complete model has been reduced to 52093 elements. The estimated reduction of computational time due to the substructuring approach is proportional to the reduction of degrees of freedom. For the substructure model on Fig. 3 the computation times were cut from 8 to approximately 4 h on a workstation with 8 CPUs (two processors with 4 cores each).

2.3. Material models

Modelling of failure and damage in the composite part of flap structure has been achieved by ABAQUS built-in progressive failure and damage model. This model uses Hashin's failure initiation criterion and accounts for the following failure modes: fibre rupture in tension, fibre buckling and kinking in compression, matrix cracking under transverse tension and shearing, matrix crushing

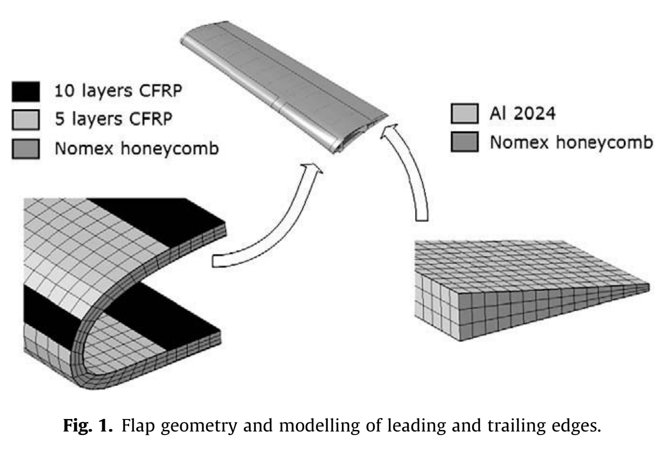

Fig. 1. Flap geometry and modelling of leading and trailing edges.