Composite Structures 109 (2014) 198–210

Contents lists available at ScienceDirect

Composite Structures

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

Finite element modeling of mechanically fastened composite-aluminum joints in aircraft structures

Zlatan Kapidžić [a,b,*], Larsgunnar Nilsson [b], Hans Ansell [a]

[a] *Saab AB, SE-581 88 Linköping, Sweden*
[b] *Division of Solid Mechanics, Linköping University, SE-581 83 Linköping, Sweden*

ARTICLE INFO

Article history:
Available online 11 November 2013

Keywords:
Bolted joints
Composite-aluminum
Finite element modeling
Hybrid wing structures

ABSTRACT

A three-dimensional, solid finite element model of a composite-aluminum single-lap bolted joint with a countersunk titanium fastener is developed. The model includes progressive damage behavior of the composite and a plasticity model for the metals. The response to static loading is compared to experimental results from the literature. It is shown that the model predicts the initiation and the development of the damage well, up to failure load. The model is used to evaluate the local force–displacement responses of a number of single-lap joints installed in a hybrid composite-aluminum wing-like structure. A structural model is made where the fasteners are represented by two-node connector elements which are assigned the force–displacement characteristics determined by local models. The behavior of the wing box is simulated for bending and twisting loads applied together with an increased temperature and the distribution of fastener forces and the progressive fastener failure is studied. It is shown that the fastener forces caused by the temperature difference are of significant magnitude and should be taken into account in the design of hybrid aircraft structures. It is concluded that, the account of the non-linear response of the joints results in a less conservative load distribution at ultimate failure load.

© 2013 Elsevier Ltd. All rights reserved.

1. Introduction

The proportion of fiber-reinforced composites in modern aircraft structures is constantly increasing due to their favorable lightweight material properties. At the same time, metals are still required in primary aircraft components e.g. when damage tolerance design is utilized. Consequently, the number of hybrid interfaces where composites and metals are joined, is growing. A common type of such interface is a shear loaded, bolted joint with a titanium bolt and carbon fiber-reinforced polymer (CFRP) and aluminum plates. This study deals with simulation of structures that contain a large number of such joints.

Bolted joints are often weak parts of the structure. It is therefore important that they are properly designed from static, fatigue and damage tolerance points of view. From the weight point of view, the joints must not be conservatively over-dimensioned. The need for an effective and accurate analysis method is from these reasons apparent. Traditionally, the development and design of joints have been based on experimental testing of specimens, which has been shown to be both expensive and time-consuming. In recent years, the use of finite element (FE) methods to simulate the behavior of composite joints has increased. In order to replace the experiments, the analysis method should be able to accurately capture a number of issues, such as: three-dimensional state of stress and strain, material behavior, bolt pretension and clamping force, bolt hole clearance, friction, secondary bending effects, contact between the surfaces etc. Another matter, that further complicates the modeling, is that there is no obvious ready-to-use material model for the failure of composites. The complex nature of CFRP failure makes it very hard to derive a universal material model with damage initiation and propagation that works well in all possible situations. Nevertheless, a large number of different criteria for the initiation and development of damage have been proposed over the years. A review with the most common methodologies for modeling of failure can be found in [1].

Another review particularly concerning the failure of bolted composite joints is presented in [2]. Since that review, a vast number of analytical studies of CFRP fastened joints has been published. They range from two-dimensional models with rigid pins representing the bolt [3–5], to three-dimensional models with or without damage development [6–15]. In these studies, bolt pretension, bolt clearance, the effect of countersunk/protruding bolt heads, different failure initiation and damage progression models, by-pass loading and implicit/explicit solution methods have been studied. Also, a large number of experimental investigations has been done, for instance in [16,17]. The level of detail in FE

---

[*] Corresponding author at: Saab AB, SE-581 88 Linköping, Sweden. Tel.: +46 (0)13183884.

E-mail address: zlatan.kapidzic@saabgroup.com (Z. Kapidžić).

0263-8223/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compstruct.2013.10.056

modeling, in the studies mentioned, requires an extensive computer power in order for the computational time to be reasonable, even for small joint installations. In structures with a very large number of fasteners, the computational and modeling burden tends to become unmanageable if no simplifications are introduced.

This has motivated several authors to develop different approaches suited for the incorporation of mechanically fastened joints into FE analysis on a structural level. In most of these methods, the fasteners are represented by structural finite elements. For example, Weyer et al. [18] used connector elements available in the program Abaqus to represent self-piercing rivets in crash analysis. The modeling included elastic, plastic and damage behavior of a single bolted metal sheet specimen. Gray and McCarhty [19] developed a method using beam elements connected to a rigid surface to represent the bolt, and shell elements to model the laminate plates of the joint. Bolt-hole clearance, bolt-torque and friction between the plates were considered, but the damage dissipation in laminates was not accounted for. The functionality of the model was demonstrated on a three-bolt single-lap joint. Another efficient approach was developed by Gray and McCarthy [20], were a user-defined element was implemented in Abaqus. The method is capable of modeling non-linear behavior and failure of composite joints based on semi-empirical approach. A joint with twenty fasteners was used for validation of the method. Ekh and Schön [21] made use of beam elements to represent both bolt and laminates and connector elements to account for bolt-hole clearance and friction. The model was suited for optimization of load transfer, but was limited to single-column joints. A two-dimensional model of a steel–concrete beam with spring elements representing the shear connections was developed by Wang and Chung [22]. A non-linear behavior of the springs, based on experimental data was utilized.

The motivation for this work comes from the intrinsic problems of hybrid structures. If materials with different thermal and mechanical properties are mixed in structures, issues that are usually absent in homogeneous structures can arise. Some examples of material properties that can differ for composite laminates and aluminum are: thermal expansion coefficients, failure and fracture mechanisms, response to different types of loading, i.e. tensile versus compressive, fatigue accumulation and scatter, impact resistance, impact residual strength, degree of anisotropy, environmental sensitivity etc. Based on these differences, composite and aluminum materials used in aircraft structures are subject to different design and airworthiness requirements. The issues that arise in the design and certification of hybrid aircraft structures as a result of this mismatch are: thermally induced loads and deformations, multiple failure modes in joints, allowance of buckling and permanent deformations, determination of factors for statistical scatter of material properties, determination of significant load states etc. These issues have to be taken into account in the design and certification of hybrid aircraft structures. The recommended practice for certification of composite assemblies is to begin analyzing and testing at a small specimen level. This way, the risks in technology can be eliminated at an early stage before moving onto testing more complex and expensive structural parts. This is known as the Building Block Approach (BBA) [23]. When it comes to hybrid constitutions this approach may not always be appropriate. As a result of the material mismatch, a hybrid joint may behave in one way when tested as a specimen and in another way when installed in a structure. Based on this argument, the analyzes and tests should be done at a structural level first in order to evaluate the true behavior of a structure. Such testing can be very expensive and a lot of effort can be saved if these tests can be done virtually, i.e. in a computer simulation. In this study, the behavior of a large winglike hybrid box is examined by FE simulations. Fatigue testing of this structure exposed to spectrum loading and an applied temperature, as well as static testing, will be performed in the near future and the results will be compared. The objective of this work is to develop a methodology for simulation of structures that contain hybrid CFRP-aluminum shear loaded bolted joints with damage and failure behavior.

The study is divided into two parts. In the first part, the local behavior of a CFRP-aluminum single bolt joint is modeled using the FE code Abaqus [24]. A three-dimensional FE model, using solid elements, is developed. For the composite plate, a progressive damage model (PDM) is implemented and aluminum and titanium parts are modeled using an elasto-plastic material behavior. The resulting joint behavior is compared to results from the experiments performed by Ireman et al. [16] in order to verify the method. In the second part, the method from the first part is used to evaluate the local behavior of several joint configuration that are present in the wing box. These behaviors are then assigned to connector elements available in Abaqus and included into the model of the wing box. Mechanical and thermal loads are applied and the redistribution of fastener loads due to joint failure is examined.

2. Modeling of composite-aluminum bolted joints

In this section, the mechanical behavior of a single-lap, shear loaded bolted joint specimen shown in Fig. 1 is assessed. The specimen was previously tested by Ireman et al. [16] where it was denoted AC6III. It contains one plate made of CFRP material HTA/6376 with ply thickness of 0.13 mm and a quasi-isotropic stacking sequence $[(\pm 45/0/90)_4]_S$. The other plate is made of aluminum AA7475-T76. A single 6 mm countersunk bolt, made of titanium Ti–6Al–4V STA is used with finger-tight pretension. The fitting tolerance between the bolt and hole is according to ISO f7/H10, which gives a clearance of 10–70 μm.

The testing with an applied shear load was performed without any lateral support, which means that secondary bending effects were present in the joint. The load–deflection response was monitored at the loading grips and the relative displacement of the plates was obtained by extensiometers attached to the sides of the specimen. The specimen was loaded until final failure occurred in a bearing failure mode. Ireman [12] also performed an FE analysis of the specimen testing but did neither include the failure behavior of the composite nor the plasticity of the metals.

2.1. Finite element modeling

In this work, FE modeling of the AC6III specimen is performed using eight-node brick elements with reduced integration, C3D8R in Abaqus. The whole mesh is shown in Fig. 2 and a closer, cut-through view of the bolt and composite plate is shown in Fig. 3. As can be seen, the bolt, the washer and the nut are considered to be one solid piece. The aluminum plate is meshed with 8 elements in the thickness direction, and in the CFRP plate each ply is represented using one element. In both plates, the mesh is refined close to the contact area between the bolt and the plates. Some mesh convergence studies, based on stresses and strains in this area are performed, but the results are not presented in this paper. For stabilization of the initial loading procedure and to eliminate rigid body modes, weak spring elements are connected to the plates and the bolt in all three directions. Furthermore, a linear spring element is connected in series with the specimen to account for the flexibility of the testing machine, as was done in [25].

In order to eliminate hourglass modes in the reduced integrated elements, Abaqus assigns an hourglass stiffness based on the

Download English Version:

https://daneshyari.com/en/article/6708264

Download Persian Version:

https://daneshyari.com/article/6708264

Daneshyari.com