Thin-Walled Structures 62 (2013) 65–74

Contents lists available at SciVerse ScienceDirect

Thin-Walled Structures

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

Numerical simulation of aluminium stiffened panels subjected to axial compression: Sensitivity analyses to initial geometrical imperfections and material properties

R.M.F. Paulo [a,*], F. Teixeira-Dias [b], R.A.F. Valente [a]

[a] GRIDS Research Group, Centre for Mechanical Technology and Automation, Department of Mechanical Engineering, University of Aveiro, Campus de Santiago 3810-193, Aveiro, Portugal

[b] GRIDS Research Group, DAPS—Division of Armour & Protection Systems, Department of Mechanical Engineering, University of Aveiro, Portugal

ARTICLE INFO

Article history:
Received 14 May 2012
Received in revised form
4 July 2012
Accepted 4 July 2012
Available online 23 September 2012

Keywords:
Integrally stiffened panels
Elasto-plastic buckling
Finite element modelling
Initial geometrical imperfections
Heat affected zone

ABSTRACT

In the present work, a set of finite element analyses (FEA) was carried out, using Abaqus to reproduce the mechanical behaviour of integrally stiffened panels when subject to longitudinal compression. Since most fabrication processes, such as welding, introduce distortions and affect the material properties, the sensitivity to these defects was assessed. Different shapes and magnitudes of the initial geometrical imperfections were tested and a high sensitivity was observed to both factors on the ultimate load. The existence of a heat affected material showed no influence on the ultimate strength of the tested panels.

1. Introduction

Stiffened panels are often the basic structural building blocks of airplanes, vessels and other structures with high requirements of strength-to-weight ratio. They typically consist of a plate with equally spaced longitudinal stiffeners on one side, and often with intermediate transverse stiffeners [1]. Large aeronautical and naval parts are primarily designed based on their longitudinal compressive strength. The structural stability of such thin-walled structures, when subjected to compressive loads, is highly dependent on the buckling strength of the structure as a whole and of each structural member.

A typical load vs. end-shortening curve, as obtained from a stiffened panel subjected to longitudinal comprehensive loads, is illustrated in Fig. 1, where distinct load–displacement regions are visible. The stiffness of a panel is reduced after buckling, as the panel enters a post-buckling regime after the ultimate load is reached, with the consequent failure or collapse [2]. After the start of buckling, a phenomenon of "mode-switching" may occur, with the panel changing the buckling patterns for increasing load levels [3,4].

Aluminium stiffened panels have been used for a long time in airplane structures, as aluminium alloys have been the primary material of choice in the aeronautical industry since the 1930s [5]. Design methods in this field were initially based on Euler's column buckling theory as well as on Timoshenko's theory on the elastic stability of plates and shells.

The use of aluminium in the construction of high speed commercial and military vessels rapidly expanded since the early 1990s, mostly due to their higher strength-to-weight ratio when compared to steel structures [6]. Naval structural design analyses have been typically simplified and experience-based [2], mainly relying on empirical methods.

In the meanwhile, specific software packages have been developed (e.g. DNV PULS, ALPS/ULPAC, Hypersizer) in order to reduce the calculation time of the ultimate load on stiffened panels. The EUROCODE 9 [8] and the US Aluminium Association Specifications [9] design codes also take into account the calculation of the ultimate strength of a stiffened panel. However, some of the previously referred methods have limitations related to the simplicity of boundary conditions that can be used, constitutive behaviours (often assumed as elastic) and geometrical configuration of the panels. Therefore, the prediction of critical loads does not always show good accuracy when compared to experimental results [2]. The use of the finite element method (FEM) can lead to higher levels of accuracy and generality, being suitable for generic case studies. Distinct FEM models, with different element

* Corresponding author.
E-mail address: ruipaulo@ua.pt (R.M.F. Paulo).

0263-8231/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.tws.2012.07.024

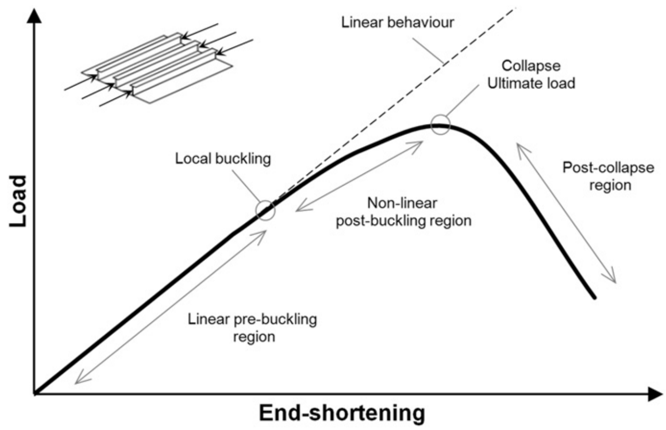

Fig. 1. Typical behaviour of a stiffened panel subjected to longitudinal compressive loads (adapted from Ref. [7]).

formulations and mesh densities were used to study the behaviour of aluminium panels in a number of works in the literature (e.g. Abaqus [10–15], ANSYS [1,16–18], LS-DYNA [18], ULSAS [18] and MSC Marc [18], to name but a few).

The manufacturing process of stiffened panels often involves welding operations necessary to join the stiffeners to the skin plates (as in build-up panels), or to join together integrally stiffened panels (ISP), that are basically modular structures including the base plate (skin) and the stiffener (stringer) in a single component directly obtained by extrusion operations. The heat added and conducted in the pieces to be joined can influence the structural efficiency of the final set, since it affects the quality of the panels in terms of: geometrical distortions, build-up of residual stresses and significant changes in the properties of the materials in the heat affected zone (HAZ) [19]. These changes are less significant for the friction stir welding (FSW) process than in the traditional welding processes, like MIG [19,20].

One main concern using the FEM in the study of the behaviour of such structures is the initial imperfection modelling. Imperfections are normally introduced into finite element models in order to provide a triggering for the initiation of buckling in the numerical simulation. Considering the use of geometrical imperfections, several authors have concluded that the shape and magnitude of the initial imperfections significantly affect the behaviour of a stiffened plate [16,18,21,22].

Different methods were used to take into consideration these initial imperfections in the analysis of aluminium stiffened panels, such as:

- previous deflection of the panel using pressure in one side of the plate [1,16,18];
- from the analysis of buckling modes [11];
- displacement of the transversal central nodes of the panel [10,14];
- displacement of the nodes based on equations (as used by Paik et al. [17]). Examples of those equations are presented and used by Zhang et al. [23] for steel panels analyses and
- displacement based on measured imperfection in the experimental panels [21,22].

In concerning the properties of the HAZ materials, their modelling properties have been presented by several authors [1,14,16,18,20,24]. They were considered in some FEM models in order to study their influence in the panels' strength behaviour. Literature reported that it can significantly reduce the ultimate strength of the panels when compared to those with original unaltered material properties [1,14,16,18].

In order to provide a comprehensive and global analysis of the behaviour of ISP under compressive loads, the present work proposes and describes a methodology and the corresponding finite element formulations and models able to reproduce the behaviour of these structures. The sensitivity of the obtained results to the shape and magnitude of the initial geometrical imperfection fields is assessed, as well as the influence of a heat-affected material in the HAZ.

2. Numerical model

2.1. Reference experimental tests

The finite element models presented in this study are based on experimental tests on the behaviour of axially compressed stiffened panels as provided by Aalberg et al. [24]. Two different types of AA6082-T6 panels were tested on this experimental work. These panels were manufactured by the Hydro Aluminium Maritime to be used in high-speed ferries (catamarans). Five single stiffener extruded sections were joined by welding for both experimental types of panels, as reproduced in the FEM models in the present work. The panels with L-shaped stiffeners were welded by MIG welding while the panels with trapezoidal closed-section were joined using a friction stir welding (FSW) technique. It should be noticed that only the short panels' configurations, as tested in Ref. [24], was reproduced for the numerical models in the present work.

In that work, two types of boundary conditions were experimentally tested along the panel's longitudinal unloaded edges: free edges and supported edges with restricted movement mainly in the normal direction of the panel plate. The loaded edge and its opposite edge were both associated to bearing support beams that allowed rotation in a direction parallel to these edges. The loaded edge was also allowed to rotate around a direction normal to the panel plate and displacement was allowed in the load direction. The edge bearing beam opposite to the loaded edge had the displacement restricted in all directions. Further information on the reference experimental test panels, test rig or experimental results can be found in the work by Aalberg et al. [24].

2.2. Model geometry and boundary conditions

Particular attention was given in the numerical simulation model to the correct reproduction of the boundary conditions and the configuration of the loading edges.

All simulations were carried out using shell elements [25] and, as a consequence, the panels are represented by their mid-section profile (the reference surface for the shell elements). The two different geometries mentioned before were considered for the numerical models, the first one corresponding to the panels with trapezoidal stiffeners (model TR) and the second representing the panels with L-shaped stiffeners (model L), as schematically shown in Fig. 2. The dimensions used in the models' transversal section were based on nominal dimension of the experimental panels. The thicknesses of the walls were based on the measured dimensions [24].

The boundary conditions that were considered in the numerical simulation models are illustrated in detail in Fig. 3, for the model TR. As shown in the figure, only half of the panel was considered, taking advantage of its symmetry in terms of load, boundary conditions and geometry. Therefore, this being the case, rotation of the loaded rigid wall was not allowed around OZ axis, as happens in the model L, where similar boundary conditions were considered but the entire panel was considered in the model (Fig. 4). For both models (TR and L) the support and non-support conditions along the unloaded edges were considered and tested.