Available online at www.sciencedirect.com

ScienceDirect

Procedia Structural Integrity 5 (2017) 713–720

Structural Integrity Procedia

www.elsevier.com/locate/procedia

2nd International Conference on Structural Integrity, ICSI 2017, 4-7 September 2017, Funchal, Madeira, Portugal

Influence of material non-linearity on load carrying mechanism and strain path in stiffened panel

Mihkel Kõrgesaar[a]*, Jani Romanoff[a], Heikki Remes[a]

[a]*Aalto University, Department of Mechanical Engineering, Puumiehenkuja 5a, 00076 Aalto, Espoo, Finland*

Abstract

This paper investigates the influence of material non-linearity on load carrying mechanism and strain path in stiffened panel. First, clamped stiffened panel with dimensions of 1.2 x 1.2 m was penetrated with rigid indenter until fracture took place. Second, panel material was characterized with standard tensile tests using flat test coupons extracted from the face sheet of the panel. Failure strain for different element lengths was calibrated using iterative state-of-the-art procedure. Numerical finite element simulations were performed using failure strain calibrated with tensile tests. Comparison of numerical and experimental force-displacement curves of panel clearly shows that this widely used approach is not sufficient for reliable element size independent numerical simulations. The reason is that failure strain scaling depends on the element size as well as stress state. The stress state in the structural component however, can considerably vary from that observed in tensile test.

Peer-review under responsibility of the Scientific Committee of ICSI 2017

Keywords: Type your keywords here, separated by semicolons ;

1. Introduction

A reliable consideration of material non-linearity and failure strain in crashworthiness analysis of large complex structures such as ship is a challenge with an increasing importance in the last decades. Ship collisions, groundings and penetration of objects through the shell plating of the ship can lead to loss of ship buoyancy and consequent

* Corresponding author. Tel.: +37253439557
E-mail address: Mihkel.korgesaar@aalto.fi

2452-3216
Peer-review under responsibility of the Scientific Committee of ICSI 2017
10.1016/j.prostr.2017.07.050

progressive flooding. The size and location of the opening defines the seriousness of the incident and thus, also the extent of the final damage. Therefore, prediction of the opening size has fundamental importance in ship safety assessment especially in regions with high traffic. In this respect, non-linear finite element simulations including ductile fracture have become a standard design tool to predict structural response for accidental loads. As the simulated structures are large, ductile fracture must be resolved with plane stress structural shell elements where element length is orders of magnitude higher than micro-scale fracture process. Moreover, ductile fracture in metals is preceded by localization of strains, i.e. necking, that makes the finite element solution mesh size dependent if no special strain gradient dependent plasticity theories are adopted (Mikkelsen, 1997). Consequently, the governing challenge in fracture modelling of large structures simplifies to correctly describing the post-necking behavior of material until point of fracture initiation with large shell elements, or in other words, *upscaling* the material behavior from simple coupon tests to large structural shell elements, see the illustration in Fig. 1a.

In general, the fracture modelling approaches adopted in the ship collision and grounding analyzes can be separated into two categories: coupled and uncoupled. In coupled approaches fracture is modelled as a process of damage accumulation within continuum, hence the coupling of constitutive model and fracture model (AbuBakar and Dow, 2013; Woelke and Abboud, 2012; Kõrgesaar and Romanoff, 2014). In the second approach fracture is considered as a sudden event when the stress or strain states of the undamaged continuum reach a critical level, beyond which element is considered as failed and removed from the analysis. The latter approach is more common in the analysis of ship structures and thus, is also adopted here. In this context one of the key parameters is the element failure strain. The term "element failure strain" is pertinent to analysis of large structures where element size acts as an averaging window for local strains that exhibit gradients dependent on the amount of necking. Commonly, the failure strain for different element lengths is determined using hybrid numerical-experimental approach (Simonsen and Lauridsen, 2000; Alsos et al., 2009). First, the standard tensile test with a dog-bone specimen is conducted where the load and displacement are recorded. Thereafter, tensile test is simulated with different element lengths, whereas failure strain for each element size is iteratively changed to capture the experimental fracture initiation point. The logarithmic relation fitted to the data is called Barba's law which is then extrapolated and used in the crash analysis of large structures. As shown in Fig. 1b, the observed relatively strong mesh size dependence arises due to combination of diffuse and localized necking.

Although this approach is widely used, there is still limited understanding of the influence of the used simplifications, e.g., the difference in strain path between tensile test and real structure on fracture prediction. In order to get deeper insight to these differences, this paper presents initial results on the influence of material-non linearity on load-carrying mechanics and strain-path in stiffened panel commonly used in ship side structure. Therefore, the approach mentioned above is used to simulate the failure in stiffened panel under quasi-static indentation. The key challenges in used approach are discussed.

2. Experiments and methods

2.1. Uniaxial tension test and simulations

Material of the tested stiffened panel is a standard structural steel S235JR commonly used in shipbuilding. To characterize the material behavior quasi-static tensile tests were performed with 3 mm thick dog-bone specimens (see Fig. 2b) using a 100 kN MTS servo-hydraulic universal testing machine. Test matrix is shown in Table 1. The loading speed was 2 mm/min. During testing, the force and displacement were recorded and these are shown in Fig. 2(a).

The equivalent stress - equivalent plastic strain curve was determined with iterative numerical approach; for results see Table 2. Material parameters were modified in the numerical finite element (FE) simulation until adequate correspondence was reached with measured force-displacement curve (see e.g. Dunand and Mohr, 2010; Luo and Wierzbicki, 2010). The elastic properties of the materials were described by a Young's modulus of E = 200 GPa, Poisson ratio of $\nu = 0.3$, and density of 7850 kg/m^3. von Mises yield criterion was used in the simulations assuming associated plastic flow and isotropic hardening. Strain hardening was described by a three-term Voce (1948) type of saturation model with yield stress of σ_0 and hardening parameters Q_i, C_i (i = 1~3) and n, see Table 2: