Marine Structures 59 (2018) 142–157

Contents lists available at ScienceDirect

Marine Structures

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

MARSTRUCT benchmark study on nonlinear FE simulation of an experiment of an indenter impact with a ship side-shell structure

Jonas W. Ringsberg[a,*], Jörgen Amdahl[b], Bai Qiao Chen[c], Sang-Rai Cho[d], Sören Ehlers[e], Zhiqiang Hu[f], Jan M. Kubiczek[e], Mihkel Kõrgesaar[g], Bin Liu[h], Janis N. Marinatos[i], Karol Niklas[j], Joško Parunov[k], Bruce W.T. Quinton[l], Smiljko Rudan[k], Manolis Samuelides[i], Carlos Guedes Soares[c], Kristjan Tabri[m], Richard Villavicencio[n], Yasuhira Yamada[o], Zhaolong Yu[b], Shengming Zhang[n]

[a] *Department of Mechanics and Maritime Sciences, Chalmers University of Technology, Gothenburg, Sweden*
[b] *Department of Marine Technology/Centre for Autonomous Marine Operations and Systems (AMOS), Norwegian University of Science and Technology, Trondheim, Norway*
[c] *Centre for Marine Technology and Ocean Engineering (CENTEC), Instituto Superior Técnico, Universidade de Lisboa, Lisbon, Portugal*
[d] *Department of Ship Building and Marine Engineering, University of Ulsan, Ulsan, Republic of Korea*
[e] *Institute for Ship Structural Design and Analysis, Hamburg University of Technology, Hamburg, Germany*
[f] *School of Engineering, Newcastle University, Newcastle upon Tyne, UK*
[g] *Department of Mechanical Engineering, Aalto University, Espoo, Finland*
[h] *School of Transportation, Wuhan University of Technology, Wuhan, China*
[i] *Marine Structures Division, School of Naval Architecture and Marine Engineering, National Technical University of Athens, Athens, Greece*
[j] *Faculty of Ocean Engineering and Ship Technology, Gdansk University of Technology, Gdansk, Poland*
[k] *Department of Naval Engineering and Marine Technology, University of Zagreb, Zagreb, Croatia*
[l] *Faculty of Engineering and Applied Science, Memorial University of Newfoundland, St. John's, Newfoundland, Canada*
[m] *School of Engineering, Department of Civil Engineering and Architecture, Tallinn University of Technology, Tallinn, Estonia*
[n] *Global Technology Centre, Lloyd's Register EMEA, Southampton, United Kingdom*
[o] *National Institute of Maritime, Port and Aviation Technology, Tokyo, Japan*

ARTICLE INFO

Keywords:
Benchmark study
Experiment
Failure criteria
Failure modes
Finite element analysis
Ship collision

ABSTRACT

This paper presents a benchmark study on collision simulations that was initiated by the MARSTRUCT Virtual Institute. The objective was to compare assumptions, finite element models, modelling techniques and experiences between established researchers within the field. Fifteen research groups world-wide participated in the study. An experiment involving a rigid indenter penetrating a ship-like side structure was used as the case study. A description of how the experiment was performed, the geometry model of it, and material properties were distributed to the participants prior to their simulations. The paper presents the results obtained from the fifteen FE simulations and the experiment. It presents a comparison of, among other factors, the reaction force versus the indenter displacement, internal energy absorbed by the structure versus the indenter displacement, and analyses of the participants' ability to predict failure modes and events that were observed in the experiment. The outcome of the study is a discussion and recommendations regarding mesh size, failure criteria and damage models, interpretation of material data and how they are used in a constitutive material model, and finally, uncertainties in general.

---

[*] Corresponding author.
E-mail address: Jonas.Ringsberg@chalmers.se (J.W. Ringsberg).

https://doi.org/10.1016/j.marstruc.2018.01.010
Received 9 October 2017; Received in revised form 10 January 2018; Accepted 22 January 2018
0951-8339/ Crown Copyright © 2018 Published by Elsevier Ltd. All rights reserved.

Nomenclature

List of abbreviations

DOF	Degrees of Freedom
FE	Finite Element
FEA	Finite Element Analysis
FEM	Finite Element Method

List of symbols

B	Breadth of the test object [m]
E	Elastic modulus [Pa]
H	Height of the test object [m]
K	Hardening coefficient [Pa]
l	Element length in FE model [m]
L	Length of the test object [m]
n	Hardening exponent [−]
t	Element thickness in the FE model [m]
W	Width of the test object [m]
x, y, z	Coordinates [m]
ε_{f}	Fracture strain [−]
ε_{n}	Necking strain [−]
$\varepsilon_{\mathrm{true}}$	True strain [−]
σ_{true}	True stress [Pa]
σ_{y}	Yield stress [Pa]

1. Introduction

The impact resistance of a ship or offshore structure subjected to collision can be quantified by the energy absorbed by the structure during its deformation and fracture. Explicit finite element (FE) analysis is an established method that is used to simulate collisions and to analyse the crashworthiness of the structures involved. Recent advancements in computational capacity, resources and commercial FE software have reduced the computation time and made it easier for engineers and researchers to carry out crashworthiness studies of large-scale and complex marine structures. One should, however, not underestimate the challenges involved in ensuring realistic and reliable results from these types of simulations and studies, which require in-depth understanding of simulation model and analysis procedure, including for example, choice of element type, mesh resolution, modelling and representation of material characteristics (elastic-plastic deformation, failure criterion, damage modelling, strain-rate effects, etc.), contact conditions, boundary conditions and numerical setting related to the FE software used and its solver.

It is important to continuously strive for model validation to ensure that the results from numerical simulations and predictions can form a solid basis for decision making in, e.g., the design of safe ships. Several investigations in the literature have shown how challenging it is to capture the sequential degradation and failure of a collided structure because of plastic deformation, fracture of its parts (sheets, stiffeners) and collapse by buckling (web frames, stiffeners). Ehlers et al. [1] presented FE simulations of the collision response of three different ship side structures. The study focused on determining the influence of different failure criteria and mesh sensitivity on the force-penetration results. Recommendations for element size and element length to thickness ratio were suggested, together with the failure criteria utilised in the study. Hogström et al. [2] presented an experimental and numerical study of the effects of length scale and strain state on the necking and fracture behaviours in sheet metals. They applied the results reported by Hogström et al. [3] in a parameter study of the material characteristics' influence on damage stability analyses of a collided ship. Recommendations on how ship collision analyses should be established were proposed, considering, among other factors, the dispersion of the material, failure criterion, modelling of striking bow section, friction and contact conditions, collision angle and striking ship speed.

Ship side and bottom structures are mainly composed of stiffened panels and web girders, whose impact strengths have been investigated extensively [4,5]. Liu and Guedes Soares [6] and Liu et al. [7] have presented methods that combine external dynamics and internal mechanics in ship collisions for design appraisal assessments. Recently, there has been growing interest in the internal mechanics of structures impacted by a bulbous bow structure. Yu and Amdahl [8] proposed a full 6 DOF coupled simulation procedure for collision and grounding accidents, using an approximation of the hydrodynamic loads that captures the major effects of the fluid. Zhang and Pedersen [9] re-examined the validity and accuracy of the simplified method proposed by Pedersen and Zhang [10] for ship collision damage analysis in ship design assessments by comprehensive validations with experimental results from the public domain. It was concluded that the damaged spaces in heavy collisions can be assumed to be the same as the contour of the penetrating rigid bow of the striking vessel, and the rupture strain of the materials can be taken from standard coupon tensile tests.

Over the past decades, numerical analyses such as finite element analysis (FEA) have been widely used to investigate the structural performance of ships and offshore structures (see, e.g., Paik [11] and Ehlers et al. [12]) because of the development of high-performance computers. However, the definitions of certain parameters in FEA strongly influence the calculated structural responses. Kõrgesaar et al. [13] presented ship collision simulations considering four different fracture criteria, three different mesh densities and two different material models in ABAQUS. The metal's failure criterion was also examined by Liu et al. [14] and further examined and extended in Ref. [15]. Liu et al. [16] developed a numerical method for simulating structure impact problems and studied the effects of mesh sizes on failure strain and the impact response of a stiffened plate impacted laterally by a spherical indenter using LS-DYNA.

Marinatos and Samuelides [17] presented a numerical method for modelling of a steel's material curve and rupture criterion, considering the effects of mesh size and strain-rate effect. Subsequently, they applied the proposed method in numerical simulations of eighteen indentation tests conducted by different research groups using three different failure criteria. They concluded that realistic simulation of the tests and consistency in terms of the representation of the deformation patterns and the estimation of the absorbed energy is achieved with the criterion based on the maximum equivalent plastic strain with a cut-off value for triaxialities

Download English Version:

https://daneshyari.com/en/article/6757983

Download Persian Version:

https://daneshyari.com/article/6757983

Daneshyari.com