Ringsberg J, Amdahl J, Chen B, Cho S, Ehlers S, Hu Z, Körgesaar M, Liu B,
Niklas K, Parunov J, Samuelides M, Soares C, Tabri K, Quinton B,
Yamada Y, Zhang S.

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

Marine Structures 2018, 59, 142-157.

Copyright:

© 2018. This manuscript version is made available under the [CC-BY-NC-ND 4.0 license](#)

DOI link to article:

<https://doi.org/10.1016/j.marstruc.2018.01.010>

Date deposited:

25/01/2018

Embargo release date:

06 February 2019

This work is licensed under a
[Creative Commons Attribution-NonCommercial-NoDerivatives 4.0 International licence](#)

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

Jonas W. Ringsberg^{1,‡}, Jörgen Amdahl², Bai Qiao Chen³, Sang-Rai Cho⁴, Sören Ehlers⁵, Zhiqiang Hu⁶, Mihkel Kõrgesaar⁷, Bin Liu⁸, Karol Niklas⁹, Joško Parunov¹⁰, Manolis Samuelides¹¹, Carlos Guedes Soares³, Kristjan Tabri¹², Bruce Quinton¹³, Yasuhira Yamada¹⁴, Shengming Zhang¹⁵

¹ Department of Mechanics and Maritime Sciences, Chalmers University of Technology, Gothenburg, Sweden

² NTNU, Norway

³ University of Lisbon, Portugal

⁴ University of Ulsan, Korea

⁵ Hamburg University of Technology, Germany

⁶ School of Engineering, Newcastle University, Newcastle upon Tyne, UK

⁷ Aalto University, Finland

⁸ Wuhan University of Technology, P.R. China

⁹ Gdansk University of Technology, Poland

¹⁰ University of Zagreb, Croatia

¹¹ National Technical University of Athens, Greece

¹² Tallinn University of Technology, Estonia

¹³ Memorial University of Newfoundland, Canada

¹⁴ Institute of Maritime, Port and Aviation Technology, Japan

¹⁵ Lloyd's Register Group Limited, UK

[‡] **Corresponding author:**

E-mail: Jonas.Ringsberg@chalmers.se; Tel: +46-(0)31-7721489

ABSTRACT

Explicit finite element (FE) analysis is an established method which is used to simulate ship collisions and analyse the crashworthiness of the structures involved. The recent advancement of computational capacity, resources and commercial FE software have reduced the computation time and made it easy for engineers and researchers to carry out crashworthiness studies of large-scale and complex marine structures. This paper presents a benchmark study on collision simulations and it was initiated by the MARSTRUCT Virtual Institute. The objective was to compare assumptions, models, modelling techniques and experiences between established researchers within the field. Fifteen research groups world-wide participated in the study. An experiment of an indenter that

penetrates a ship-side structure was used as the case study. A description of how the experiment was performed, a geometry model of it, and material properties, were distributed to the participants prior to their simulations. The paper presents the results from the fifteen FE simulations and the experiment. It presents a comparison of among others 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 element size, failure criterion and damage models, interpretation of material data and how it is used in a constitutive material model, and finally, uncertainties in general.

Keywords: benchmark study; ship collision; finite element analysis; experiment; failure criteria; failure modes;.

Nomenclature

List of abbreviations

FE	Finite Element
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 [-]
$\varepsilon_{\text{true}}$	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 which is used to simulate collisions, analyse various collision scenarios and the crashworthiness of the structures involved. The recent advancement of computational capacity, resources and commercial FE software have reduced the computation time and made it easy for engineers and researchers to carry out crashworthiness studies of large-scale and complex marine structures. One should, however, not underestimate the challenge that realistic and reliable results from this type of simulations and studies often require in-depth understanding of factors which are coupled in the simulation model and analysis procedure, for example, choice of element type, mesh resolution, modelling and representation of material characteristics (elastic-plastic deformation, failure criterion, damage modelling, element size, 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 and verification to ensure that the results from numerical simulations and predictions can form 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 due to 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 from 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 which were 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 in 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 set up were proposed considering among others the dispersion of the material, failure criterion, modelling of striking bow section, friction and contact conditions, collision angle and striking ship speed.

Samuelides M. [20] stated that it is essential to integrate the criterion in the FE software to simulate the propagation of rupture, because the output of rupture is subsequently used for the calculation of oil outflow, the time to capsize and the ultimate strength prediction. Normally, rupture is described

reasonably as a failure strain that is a function of the stress state and loading rate, and the simplest approach to achieving this is to select a single uniform strain value at which the local element fails. On that basis, one recent failure criteria includes the combined effect of mesh size and stress triaxiality on the failure strain was proposed by Walters [21]. The reasoning behind such scaling framework is that mesh size dependency of the FE solution depends on the amount of strain localization, which varies depending on the stress state [22]. Kõrgesaar and Kujala [23] proved that the approach based on comparison with available experimental data, the force-displacement curves of smaller panels as well as large-scale collision experiments. In addition, the effect of bending on mesh size sensitivity of the analysis was discussed in Storheim et al. [24].

2. Objective, description of benchmark study

Benchmark studies have an important purpose to fulfil when comparing different research groups' skills, best practices, assumptions and "tradition" how to design numerical models, simulate and analyse e.g. the structural response of a complex ship or offshore structure subjected to an impact load. Even if modelling guidelines and best practices are available, there are always sources of errors and uncertainties which lead to scatter in the simulation results. Benchmark studies help us to compare, learn from each other, and systematically identify issues that require improvements and sometimes new guidelines. Some important questions to discuss and try to answer in these studies are how to judge how large scatter in the results that can be accepted, which indicators or criteria that should be used in the assessment, and probably most important of all, communicate all lessons learnt which can lead to improvements and revised best practices.

The objective with this investigation was to present a benchmark study on the participants' ability, expertise and recommendations how to design FE models for collision simulations. It is expected obtain valuable summaries and experiences with the comparisons between simulations outcomes with that of model test. A reference experiment where an indenter penetrates a ship-like structure was used in a case study. It was designed to be similar to a realistic case when a striking ship's bulb penetrates the side-shell structure of a struck ship during a collision scenario. Figure 1 presents a schematic and a photograph of the experiment with the double-hull side-shell structure which is penetrated by a solid half-sphere. Measurements from the experiment and stress-strain data from uniaxial tensile tests of the steel material in the structure was made available to the participants through previous work reported in Karlsson et al. [6].

Fig. 1. (a) Geometry description of the side-shell structure and the indenter used in the benchmark study, and **(b)** a photograph of the experimental set-up.

Fifteen researchers world-wide participated this benchmark study. All of them are active within the research area collision and grounding of ship and offshore structures, and they have published numerous scientific papers on the topic during the years. The majority of the participants are active in both the International Ship and Offshore Structure Committee (ISSC) [4] and in the MARSTRUCT Virtual Institute [5] which coordinated the benchmark study through the lead author of this paper. The participants received the same information, instructions, data and files prior to the start of the study:

- The geometry model of the set-up: the side-shell structure, the indenter, and a reinforcing frame for the boundary conditions and control of the failure modes of the structure; see Section 2 for more details.
- The dimensions of all parts of the structure.
- The stress-strain curve from uniaxial tensile tests of the material in the side-shell structure.
- Coefficients from a curve fit of the stress-strain curve.
- Clear definitions of the boundary conditions.
- Definition of the contact point and conditions between the indenter and the top of side-shell structure.
- Material properties of the rigid indenter.
- Description of how the experiment was carried out: displacement-controlled, load rate and when the experiment was terminated.

The participants submitted individual reports of their results and recommendations how to design the FE model and perform the numerical simulation of the experiment. The information in these reports were compiled and are presented in this study:

- Detailed description of the FE model and all assumptions which were made, e.g. choice of finite element type, and mesh resolution of the different parts of the structure.
- The FE software and version that was used.
- Modelling of the boundary conditions in the experiment.
- Modelling of the contact and loading conditions in the experiment.
- Choice of constitutive material model to simulate the elastic-plastic behaviour of the material, which material data that was used in the model, and if special consideration was taken to strain-rate effects or not in the model.
- The choice of failure model/criterion that was used in the FE analysis together with used and assumed properties.
- Other models and assumptions made, such as Barba's law [7] for element dimensions' influence on the value of the fracture strain.

Three types of results were reported by each participant: (1) the reaction force-indenter displacement curve, (2) the internal energy-indenter displacement curve, and (3) a table which presents the indenter displacement value where a structural member failed/fractured, buckled, etc. in the FE simulation. These results were compared between the participants, the results from the experiment, and discussed in relation to how the different FE models were designed and the simulations were performed. Section 3 of this paper gives a brief summary of the reference experiment. In Section 4, a summary of the fifteen FE models is presented followed by a comparison of results in Section 5 from the FE simulations and the experiment. The section includes a discussion of the results and suggests recommendations for how this type of FE simulation with regard to FE model parameters, constitutive material model and failure criteria should be carried out. The conclusions of the study are presented in Section 6.

3. Reference experiment

The ship-like structure test geometry designed by Karlsson et al. [6] was used as the reference experiment. It resembles a typical double-hull side-shell structure (hereafter referred to as the test object) of a ship subjected to collision load where the bulb of the striking ship penetrates the structure. To fit the test object to the testing machine, it was scaled to a third of the size of a similar full-scale ship structure. This section gives a brief description of the design and how the experiment was carried out to make the presentation of the benchmark study complete. Detailed description and analyses of the experiment are presented in Hogström et al. [3] and in Karlsson et al. [6].

3.1 Description of the test object, boundary and contact conditions

The test object consisted of one outer and one inner side-shell, web/stringer sheets, web/stringer beams and stiffeners in the form of L-profiles. The global dimensions $L \times W \times H$ of the structure were 1500 mm $\times$ 1090 mm $\times$ 300 mm and the sheet thickness was between 3 mm for the thinnest and 5 mm for the thickest structural elements. In order to accomplish well-defined boundary conditions, a reinforcing rigid frame was designed around the structure. Figure 2 shows the geometry and the dimensions without the reinforcing frame.

Fig. 2. Dimensions of the test object; from Karlsson et al. [6].

The reinforcing frame was designed and welded around the test object along its edges to create clamped boundary conditions, and to ensure well-controlled failure modes of the structure. The lower part of it was welded to a rigid fixture. Four displacement transducers were positioned in two directions at the supporting frame and fixture. They measured the frame's deformation to make sure that the fixture's deformation during the tests was negligible; see Fig. 3 for the experimental set-up where two of the force transducers are seen in the front and to the left of the test object.

Fig. 3. Photograph of the test object in the test rig with the indenter (half-sphere), the reinforcing frame welded to the rigid fixture, and the displacement transducers.

The test object was manufactured of the K240-Z shipbuilding mild steel. The indenter was made as a solid (rigid) half sphere with radius 135 mm and made of the SS2541 steel. Friction tests without lubrication were carried out by Karlsson et al. [6], who showed that the kinematic friction coefficient was 0.23 ± 0.01 for the current contact conditions.

3.2 Test procedure and measurements

The test object was mounted in a press machine with 20 MN load capacity. To relax the residual stresses caused by welding of the test object's sheets to the frame before the test, the indenter was pushed perpendicular against the structure followed by unloading, at low speed ten times. The magnitude of the load in this loading sequence was within the elastic region of the material.

The indenter penetrated the structure with a constant displacement rate of 4 mm/s. The collision point was in the centre of the sheet, see Fig. 3, and the loading direction was perpendicular to the upper sheet's surface. The experiment was interrupted when the lower sheet was fully penetrated by the indenter. Throughout the experiment, the resultant force in the load cell, the position of the indenter, and the displacement transducers on the rigid frame, were monitored and collected. The total calculated measurement uncertainty for the maximum force was less than 1%. The reported uncertainty corresponds to an approximate 95% confidence interval around the measured value; see Karlsson et al. [6] for details.

4. Finite element models and analyses of the experiment

The participants of the benchmark study made their FE models by using the geometry file that defined the geometries of the test object, the indenter and the reinforcing frame. None of the welds were modelled in the FE models. The indenter was modelled as a rigid body and it was allowed to move only in the direction perpendicular to the upper sheet with a constant displacement rate. Figure 4 shows an example of an FE model with and without the reinforcing frame; the rigid indenter is not shown in the figure. Table 1 presents a summary of the participants' different FE model definitions and parameters. Two FE solvers were used, Abaqus [8] and LS-DYNA [9], where all modelling details in the current study referring to these solvers can be found in their references, respectively.

Fig. 4. Example of an FE model of the test object:

(a) with the reinforcing frame, and (b) without the reinforcing frame.

Table 1. Summary of the participants' FE model definitions and parameters.

ID	FE solver	Element type	Integration: reduced (R) or full (F)	Element size [mm]	Indenter speed [m/s]	Friction coefficient	Reinforcing frame in the model: Yes/No
1	Abaqus/Explicit v6.13-3	S4R; hourglass control	R	Sheets: 10 Other members: 9.2-9.8	1.0	0.23	Yes
2	Abaqus/Explicit v6.13-3	S4R; hourglass control	R	15	1.0	0.23	Yes
3	Abaqus/Explicit v6.13-4	S4R; hourglass control	R	15	3.0	0.23	Yes
4	Abaqus/Explicit v6.13-3	S4R; hourglass control	R	Upper sheet: 15 Lower sheet: 30	10.0	0.23	Yes
5	LS-DYNA v9.71	FE type 16	F	10	0.50	0.23	No
6	LS-DYNA v9.71, smp d R7.1.1	FE type 16	F	15	5.0	0.23	Yes
7	LS-DYNA v9.71, smp d R8.0.0	Belytschko-Lin- Tsay elements	R	10	2.0	0.23	Yes
8	LS-DYNA v9.71	Belytschko-Lin- Tsay elements	R	15	3.0	0.23	Yes
9	LS-DYNA v9.71	Hughes-Liu (HL) shell elements	R	9	0.50	0.23	Yes
10	LS-DYNA v9.71, smp d R7.0.0	Belytschko-Lin- Tsay elements	R	10	0.45	0.23	Yes
11	LS-DYNA v9.71, R7.0.0 double precision	Belytschko-Lin- Tsay elements	R	10	2.0	0.23	Yes
12	LS-DYNA v9.71, smp d R7.1.2	Belytschko-Lin- Tsay elements	R	Sheets: 10 Stiffener web: 12	0.01	0.23	Yes
13	LS-DYNA v9.71	Belytschko-Lin- Tsay elements	F	20	2.0	0.23	No
14	LS-DYNA v9.71	Belytschko-Lin- Tsay elements	R	15	2.2	0.23	Yes
15	LS-DYNA v9.71 R7.1.1	Belytschko-Lin- Tsay elements	R	10	1.0	0.23	Yes

The mesh size of the FE models was determined by convergence analysis. All participants used four-node shell elements with five section points through the thickness. The element size in Table 1 is the primary size of the elements in the parts of the structure that undergo failure and fracture under the numerical simulation. Recommended practice prescribes that the element length to thickness ratio, l/t , should be around 5. It should be noted that l/t varies from 2 to 6 among the FE models in Table 1.

The users of Abaqus/Explicit used the “general contact conditions” criterion to define the contact conditions together with the coefficient of frictions presented in Table 1. This contact criterion enforces contact constraints using a penalty contact method, which searches for node-into-face and edge-into-edge penetrations. Similarly, the users of LS-DYNA used the coefficient of frictions in Table 1 but two different contact criteria: “automatic surface-to-surface” in the contact between the indenter and the test object, and “automatic single surface” in the contacts between other structural components. Further, it should be noted that in order to save computation time, the indenter speeds were often much higher in the FE analyses compared to the physical experiment. The material properties did not include any effects from high loading rates or strain-rate effect; see further on in this section.

The indenter was assumed rigid with properties of Young’s modulus 206 GPa, Poisson’s ratio 0.3 and density 7,850 kg/m³. The test object and reinforcing frame were originally manufactured of the K240-Z shipbuilding mild steel with a density of 7,850 kg/m³ and Poisson’s ratio 0.3. The properties of this material were based on a few tensile tests which were carried out by and presented in Karlsson et al. [6]. The results were, however, not sufficient for detailed modelling and calibration of material parameters for different types of failure and fracture criteria. Therefore, Hogström et al. [3] carried out an in-depth investigation of material properties and parameters for the K240-Z shipbuilding mild steel and the almost similar NVA shipbuilding mild steel for which additional test results were available and more tests were carried out. It was found that the K240-Z and NVA shipbuilding steels had similar properties. Hence, the latter was used in the current study since more material parameters could be provided to the participants of the benchmark study.

Figure 5 presents the stress-strain curve from uniaxial tensile tests of the NVA shipbuilding mild steel. The participants received the raw data from the experiment. The isotropic hardening of the inelastic stress-strain relation follows the power law in Eq. (1). The values of the material parameters that describe the non-linear material behaviour were calculated by curve fit and shared with the

participants: yield stress, $\sigma_y = 290$ MPa; hardening coefficient, $K = 616$ MPa; hardening exponent, $n = 0.21$; necking strain, $\epsilon_n = 21\%$; and fracture strain, $\epsilon_f = 26\%$; the Young's modulus was 206 GPa.

$$\sigma_{\text{true}} = K(\epsilon_{\text{true}})^n \quad (1)$$

Fig. 5. True Stress-strain relationship for the NVA shipbuilding mild steel.

Table 2 presents a summary of the constitutive material models and their parameters for each FE model. In all of the cases, the material was represented by a nonlinear elastic-plastic constitutive material model with isotropic hardening. Because the physical experiment was carried out at low speed, the influence from strain rate effects was considered negligible, i.e. it was disregarded in the analyses. The summary shows that the participants used the provided information in different ways more than the two possibilities that were suggested, either using the raw data from the uniaxial tensile test, or, the material properties that can be used in the power law for isotropic hardening of the inelastic stress-strain.

Table 2. Constitutive material models and material parameters for the test object and the reinforcing frame.

ID	Constitutive material model	Material	Young's modulus, E [GPa]	Yield stress, σ_y [MPa]	Stress-strain relationship: curve from test or inelastic power law (K and n)	Comments
1	Isotropic hardening, inelastic Swift power law with a yield plateau [10]	NVA mild steel	206	310	$K = 700$ MPa $n = 0.195$	Own curve fit to curve from test.
2	Isotropic hardening,	NVA mild	206	290	$K = 616$ MPa	

	inelastic power law in Eq. (1)	steel			$n = 0.21$	
3	Isotropic hardening, inelastic power law in Eq. (1)	NVA mild steel	206	290	$K = 616 \text{ MPa}$ $n = 0.21$	
4	Modified Ludwik's constitutive equation [11]	NVA mild steel	206	310.5	$K = 406 \text{ MPa}$ $n = 0.468$	Own curve fit to curve from test.
5	Mat.024-piecewise linear plasticity	NVA mild steel	206	290	Curve from test.	
6	Mat.024-piecewise linear plasticity	NVA mild steel	206	290	Curve from test.	
7	Mat.123-modified piecewise linear plasticity	NVA mild steel	206	325	Curve from test.	Own fit of the yield stress.
8	Mat.123-modified piecewise linear plasticity	NVA mild steel	206	290	$K = 616 \text{ MPa}$ $n = 0.21$	
9	Mat.024-piecewise linear plasticity	NVA mild steel	206	290	Curve from test.	
10	Isotropic hardening, inelastic power law in Eq. (1)	NVA mild steel	206	290	$K = 616 \text{ MPa}$ $n = 0.21$	
11	Isotropic hardening, inelastic Hollomon power law with a yield plateau [12]	NVA mild steel	206	310	$K = 720 \text{ MPa}$ $n = 0.21$	Own curve fit to curve from test.
12	Mat.024-piecewise linear plasticity	NVA mild steel	206	310	Curve from test.	Own fit of the yield stress.
13	Mat.024-piecewise linear plasticity	K240-Z mild steel	206	235	$K = 658 \text{ MPa}$ $n = 0.194$	Own curve fit to curve from tests in Karlsson et al. [6].
14	Mat.024-piecewise linear plasticity	NVA mild steel	210	290	$K = 616 \text{ MPa}$ $n = 0.21$	
15	Mat.024-piecewise linear plasticity	NVA mild steel	206	290	Curve from test.	

One large difference between the FE models was related to which failure criterion that was preferred and how damage was modelled. The majority of the participants used a failure criterion which does not consider for a stiffness degradation after the necking point. Few participants used a multiple damage criterion which separates the damage and failure process into damage initiation (from the strain at the yield stress to the strain at the necking point) and damage evolution (from the strain at the necking point to the strain at the fracture point) and thereby accounts for a stiffness degradation after the necking point according to constitutive mechanics principles. Table 3 presents a summary of the failure criteria and the damage models that were used in the FE models. All participants made a check/correction of the element size's influence on the value of the fracture strain. It is stated in the comments if Barba's law [7] or any other model was implemented in the FE model to allow for the fracture strain's dependence on the element size and thickness in different parts of the FE model.

Table 3. Summary of the preferred failure criterion for each FE model, how damage was modelled and values of relevant and used material parameters.

ID	Failure criterion	Clarification of the failure criterion and damage models	Necking strain, ε_n [%]	Fracture strain, ε_f [%]	Comments
1	Multiple damage criterion: initiation and evolution.	Damage initiation was an own developed VUMAT, followed by an own developed damage evolution model [13].	0.21	0.26	A plastic fracture strain scaling was used based on element size and stress triaxiality, see [13]. Needs calibration of material parameters.
2	Multiple damage criterion: initiation and evolution.	Damage initiation was modelled using the shear criterion, damage evolution was modelled using a bilinear damage evolution model.	0.21	0.26	See [2,8] for details regarding the damage initiation and evolution models. Barba's law [7] is used in the FE model.
3	Shear failure criterion.	Own shear criterion in a VUMAT, see [14] for details. No separate damage evolution model after necking.		0.26	Allows for the influence of element size and thickness on the fracture strain value in the FE model according to a model in [14].
4	Shear failure criterion.	No separate damage evolution model after necking.		0.28	Allows for the influence of element size and thickness on the fracture strain value in the FE model according to a model in [15].
5	Effective plastic strain.	No separate damage evolution model after necking.		0.26	
6	Effective plastic strain.	No separate damage evolution model after necking.		0.26	
7	Effective plastic strain.	No separate damage evolution model after necking.		0.525	The fracture strain was calculated according to [16] and considered the influence from mesh size. Allows for the influence of element size on the fracture strain value in the FE model.
8	Effective plastic strain.	No separate damage evolution model after necking.		0.45	The fracture strain was calculated according to a model in [17]. Allows for the influence of element size and thickness on the fracture strain value in the FE model.
9	Effective plastic strain.	No separate damage evolution model after necking.		0.35	The value of the fracture strain was revised using Barba's law [7] to match the mesh size of the FE model.
10	Effective plastic strain.	No separate damage evolution model after necking.		0.43	The value of the fracture strain was studied in a parametric study (mesh size was one of the parameters) before one recommended and final

					value was decided.
11	The BWH (Bressan-Williams-Hill) instability criterion in [18].	The criterion was used without the post-necking damage model presented in [19].	0.21		The fracture strain is not used in the failure criterion.
12	The Germanischer Lloyd (GL) criterion based on thru thickness plastic strain [1].	No separate damage evolution model after necking.		0.26	
13	Effective plastic strain.	No separate damage evolution model after necking.		0.35	The value of the fracture strain was studied in a parametric study (mesh size was one of the parameters) before one recommended and final value was decided.
14	Effective plastic strain.	No separate damage evolution model after necking.		0.39	The value of the fracture strain was studied in a parametric study (mesh size was one of the parameters) before one recommended and final value was decided.
15	Effective plastic strain.	No separate damage evolution model after necking.		0.315	Barba's law [7] is used in the FE model.

5. Results and discussion

The results from the FE simulations of the reference experiment are presented, compared and discussed with regard to the resultant vertical force versus indenter displacement (Section 5.1), the internal energy versus indenter displacement (Section 5.2), analysis of deformations and failure modes (Section 5.3), and a discussion (Section 5.4). Like in the experiment, the FE simulations were ran until an indenter displacement of 0.5 m was reached. This indenter displacement corresponds to a full penetration of the indenter through both of the sheets of the test object. It should be noted that the registered signals from the four displacement transducers showed that the reinforcing frame was perfectly rigid throughout the experiment.

5.1 Resultant vertical force versus indenter displacement

Figure 6 shows the resultant vertical force of the indenter versus its vertical displacement. The origin for the measurement of the displacement was on the upper surface of the upper sheet of the structure. The penetration of the upper sheet is depicted by the first peak, and penetration of the lower sheet by the second peak; see Section 5.3 for a detailed analysis of the deformations and failure modes.

Overall, the results from the FE simulations capture the trend and show good agreement with the experiment. There is minor scatter between the FE simulations, and the result from the experiment is in the middle of all the curves, at least until the indenter displacement is around 0.35 m. After that, the majority of the FE simulations overestimate the force at the second peak where the penetration of the lower sheet occurs and also a small off-set in the displacement when it occurs. The results in Fig. 6(b) show that neither the users of the FE solver Abaqus, nor LS-DYNA, mimics the result from the experiment better than the other.

Fig. 6. Resultant vertical force versus displacement of the indenter. **(a)** Results from the reference experiment in [6] and the FE simulations of the benchmark study; **(b)** the same result presentation as in (a) where the black curves represent the users of the FE solver Abaqus and the red curves LS-DYNA.

5.2 Internal energy versus indenter displacement

The processes of plastic deformation and fracture of the test object are complex. In the design and analysis of crashworthiness of structures, the internal energy, which here is the energy absorbed through deformation and fracture of the structure, is an important property of its characteristics to resist external loadings. Figure 7 presents the internal energy versus the displacement of the indenter from the experiment and the FE simulations. The curves No. 2, 3, 7 and 14 show excellent agreement with the curve from the experiment, and curve No. 8 stands out since it is the only FE simulation which overestimates the internal energy. All other curves underestimate the internal energy compared to the experiment. The results in Fig. 7(b) show that neither the users of the FE solver Abaqus, nor LS-DYNA, mimics the result from the experiment better than the other.

Fig. 7. (a) Energy absorbed by the structure versus indenter displacement, and **(b)** the same result presentation as in (a) where the black curves represent the users of the FE solver Abaqus and the red curves LS-DYNA.

5.3 Analysis of deformations and failure modes

Events referring to deformation and fracture of the test object were identified to enable a comparison between the experiment and the FE models. Table 4 presents the nine events and their indenter displacements which could easily be identified in an FE simulation, but only in eight of the cases in the experiment (event 1 could not be observed). Figure 8 presents markers of the events in the curve from the experiment for the resultant vertical force versus indenter displacement. The markers refer to the mean values of the indenter displacement from FE simulations, and the observations and registered values in the experiment. Figure 9 presents snapshots from an FE simulation of the deformed structure for each event.

Table 4. Summary of the preferred failure criterion for each FE model, how damage was modelled and values of relevant and used material parameters.

Event No.	Description of the event	Experiment: indenter displacement [mm]	FE simulations: mean value of the indenter displacement [mm]	FE simulations: standard deviation of the indenter displacement [mm]	No. of participants that identified the event in their FE simulation
1	Initiation of plasticity expansion of the T-beam.	No data available.	11.6	4.7	15
2	Tripping of the T-beam.	103	89.4	18.9	14
3	Buckling of the webs of the two central L-profiles attached to the upper sheet.	132	145.2	18.8	11
4	Fracture initiation	157	157.6	19.6	15

	of the upper sheet.				
5	Folding of the webs of the two central L-profiles attached to the upper sheet.	173	169.7	39.7	14
6	T-beam off.	216	236.6	22.3	13
7	T-beam starts to contact with the L-profile attached to the lower sheet.	254	264.2	17.8	14
8	Contact of the striker with the lower sheet.	291	317.2	18.7	15
9	Fracture initiation of the lower sheet.	399	423.6	12.2	15

Fig. 8. Presentation of the events in the curve from the experiment for the resultant vertical force versus indenter displacement. The markers “o” refer to the mean values of the indenter displacement from the FE simulations, and “*” the observations in the experiment. The error bars present the standard deviation for each event from the FE simulations, see Table 4.

Fig. 9. (a) Event 1: initiation of plasticity expansion of the T-beam; (b) Event 2: tripping of the T-beam; (c) Event 3: buckling of the webs of the two central L-profiles attached to the upper sheet; (d) Event 4: fracture initiation of the upper sheet; (e) Event 5: folding of the webs of the two central L-profiles attached to the upper sheet; (f) Event 6: T-beam off; (g) Event 7: T-beam starts to contact with the L-profile attached to the lower sheet; (h) Event 8: contact of the indenter with the lower sheet; and (i) Event 9: fracture initiation of the lower sheet.

The results show rather good agreement between the simulations and the experiment for all the events except for event no. 9. Prior to this event, the test object has undergone large plastic deformation and fracture of several structural members. It is a true challenge to capture the last event of the fracture of the lower plate. It can also be so that prediction made by the FE simulations is within the range of the total uncertainty of the experiment and in particular the instant of event 9. Since results from only one experiment was available, it was not possible to find a better explanation to this deviation.

Not all of the events were observed by all participants of the benchmark study; the events 2, 3, 5, 6 and 7 were not identified by everyone. One FE model could not observe any of these 5 events, one FE model missed the events No. 3 and 6, and two other FE models missed event No. 3. Analyses of the FE models showed that the mesh resolution of the structural members (not the upper and lower sheets) and supporting structures that were involved in the deformation processes leading to these events may not be adequate. Hence, finer meshes of the FE models should have been used.

5.4 Discussion

The summaries of the FE models in the Tables 1 to 3 show that the FE models have many similarities but also differences as a result of the variety of assumptions, experiences and interpretations the participants have made according to their own best practice. Despite this, the total scatter is low, and the agreement with the results from the experiment was found to be generally very good.

Out of the fifteen participants, four used the FE software Abaqus and eleven used LS-DYNA. The results in Figs 6(b) and 7(b) show that neither the users of Abaqus, nor LS-DYNA, mimics the result from the experiment better than the other. The results from the participants No. 2, 3, 7, 10 and 14 show the best agreement with the experiment with respect to (i) the reaction force versus indenter displacement (see Fig. 6), (ii) energy versus indenter displacement (see Fig. 7), and (iii) prediction of the failure modes in Table 4. A model uncertainty analysis of how these FE models were defined showed the following:

- All of them included the reinforcing frame in the FE model.
- They used a mesh size which was either 10 or 15 mm, and the fracture strain was adjusted according to Barba's law [7] or own methodology.
- Two of them used Abaqus but with different failure criteria and damage models, three of them used LS-DYNA and the same failure criterion.

- All of them but participant No. 7 used the power law coefficient K and exponent n provided at the outset of the benchmark study; No. 7 used the curve (raw data) from the uniaxial test and made an own curve fit which also can be seen in the values of the yield stress and the fracture strain (which was also included in addition to its mesh size dependence in the method that was used).

The only difference between the Abaqus FE models No. 2 and 3 was which failure criterion and damage model that was used, and it caused only a minor difference in the results. Similarly, the difference between the LS-DYNA FE models No. 10 and 14 was the mesh size with the adjusted value of the fracture strain; the difference in the Young's modulus was assumed negligible for the current case due to the large plastic deformations and fracture processes. The conclusion is that despite these deviations in these factors, they were not sufficient to influence significantly the uncertainty in the prediction of the experiment's characteristics using these FE models.

Based on the results in this benchmark study, the authors found it difficult to pin-point which model parameters or factors that contributed the most the uncertainty in the prediction. The scatter in results was low and the participants have good experience how to design FE models and set-up this type of simulations. It could be a coincidence in a combination of small variations of model definitions that led to that five of the FE models gave somewhat less agreement with the reference experiment. In contrast to the FE models 2, 3, 7, 10 and 14:

- No. 1 and 4 used different power law data for K and n , and No. 4 used a different constitutive material model.
- No. 5 did not consider the reinforcing frame in the FE model. Participant and FE model No. 13 did not include it either, but other differences in model parameters may have cancelled the effect of not including the reinforcing frame.
- No. 11 used different power law data for K and n , and the BWH instability criterion in [18] which did not consider post-necking damage.
- No. 15 had a different combination of element size and fracture strain compared to the other FE models which basically had the same other model definitions.

The results from this benchmark study can serve as a guideline how to design FE models and set-up a numerical simulation of ship collisions. The summaries presented in the Tables 1 to 3, together with

the results in Section 5, show how different combinations, selections and variations of e.g. element size, failure criterion and damage models, material data and parameters in general, affect and contribute to the variability and uncertainty in a numerical simulation of ship collisions. It is strongly recommended to make parameter sensitivity analyses prior to “sharp” FE simulations. All the participants of this benchmark study worked according to this principle before they submitted their recommended FE model, its definitions, and simulation results.

6. Conclusions

This paper presented a benchmark study on collision simulations initiated and organised by the MARSTRUCT Virtual Institute. A comparison of the participants’ ability, expertise and recommendations how an explicit FE simulation of an experiment where an indenter penetrates a ship-like structure was presented. The experiment was designed to be similar to a realistic case when a striking ship’s bulb penetrates the side-shell structure of a struck ship during a collision. Fifteen experienced researchers within the field of ship collision and grounding world-wide participated. The results from fifteen FE models and simulations of the experiment were compared with respect to resultant force versus indenter displacement, internal energy versus indenter displacement, and failure modes of the ship-like structure.

The summary of the results from all FE simulations showed low scatter, and the agreement with the results from the experiment was generally very good. Despite some variations in the FE models with regard to e.g. element size, boundary conditions, constitutive material model and material data used, the difference in results must be considered acceptable considering the complexity in simulating this type of experiments with large plastic deformation and a number of sequential failure modes of the structure. Regardless of the choice of failure criterion and damage models used in the FE models – shear failure criterion, equivalent plastic strain criterion, the BWH criterion, a multiple damage criterion with initiation and evolution – the scatter in results between the FE simulations was acceptable and low.

The main contribution from the study is its intention to serve as a guideline how to design FE models and set-up a numerical simulation of a ship collision. The summaries of the fifteen FE models and the results from their simulations show how different combinations, selections and variations of e.g. element size, failure criterion and damage models, material data and parameters in general, affect and contribute to the variability and uncertainty in a numerical simulation of ship collisions. It is strongly recommended to make parameter sensitivity analyses prior to “sharp” FE simulations. All the

participants of this benchmark study worked according to this principle before they submitted their recommended FE model, its definitions, and simulation results.

Acknowledgments

References (to be updated and revised ...)

- [1] Ehlers, S., Broekhuijsen, J., Alsos, H.S., Biehl, F. and Tabri, K. (2008). Simulating the collision response of ship side structures: a failure criteria benchmark study. *International Shipbuilding Progress*, 55(1), pp. 127-144.
- [2] Hogström, P., Ringsberg, J.W. and Johnson, E. (2009). An experimental and numerical study of the effects of length scale and strain state on the necking and fracture behaviours in sheet metals. *International Journal of Impact Engineering*, 36(1), pp. 1194-1203.
- [3] Hogström, P. and Ringsberg, J.W. (2012). An extensive study of a ship's survivability after collision – a parameter study of material characteristics, non-linear FEA and damage stability analyses. *Marine Structures*, 27(1), pp. 1-28.
- [4] ISSC (2017) ... web-site
- [5] MARSTRUCT (2017) ... web-site
- [6] Karlsson, U., Ringsberg, J.W., Johnson, E., Hosseini, M. and Ulfvarson, U. (2009). Experimental and numerical investigation of bulb impact with a ship side-shell structure. *Marine Technology*, 46(1), pp. 16-26.
- [7] Barba's law.
- [8] Abaqus/Explicit web-site
- [9] LS-DYNA web-site
- [10] Swift (1952) power hardening law
- [11] Cho, S.-R., Choi, S.-I. and Son, S.-K. 2015. Dynamic material properties of marine steels under impact loadings. *Proc. of the 2015 World Congress on Advances in Structural Engineering and Mechanics*, ASEM15, Incheon, Korea.
- [12] J.H. Hollomon, *Trans. AIME*, 162, 268 (1945)
- [13] Körgesaar M, Romanoff J. Influence of mesh size, stress triaxiality and damage induced softening on ductile fracture of large-scale shell structures. *Mar Struct* 2014;38:1–17.
- [14] Marinatos, J.N. and Samuelides, M. (2015). Towards a unified methodology for the simulation of rupture in collision and grounding of ships. *Marine Structures*, 42, 1-32.

- [15] Cho, S.-R., Song, S.-U. Park, S.-H. and Shin, H. K. 2017. Plastic and fracture damages of double hull structures under lateral collisions. To be presented at the 6th International Conference on Marine Structures, May, 2017. Lisbon, Portugal.
- [16] Ehlers, Sören (2010), A procedure to optimize ship side structures for crashworthiness, "Proceedings of the Institution of Mechanical Engineers, Part M: Journal of Engineering for the Maritime Environment", 224,1,1-11, SAGE Publications.
- [17] Liu B, Villavicencio R, Zhang S, Guedes Soares C. 2017. A simple criterion to evaluate the rupture of materials in ship collision simulations. *Marine Structures*, Volume 54, July 2017, Pages 92-111
- [18] Alsos, H.S., et al., *Analytical and numerical analysis of sheet metal instability using a stress based criterion*. *International Journal of Solids and Structures*, 2008. **45**(7): p. 2042-2055.
- [19] Storheim, M., et al., A damage-based failure model for coarsely meshed shell structures. *International Journal of Impact Engineering*, 2015. 83: p. 59-75
- [20] Samuelides M., 2015. Recent advances and future trends in structural crashworthiness of ship structures subjected to impact loads. *Ships and Offshore Structures* Volume 10, 2015, pp. 488-497
- [21] Walters CL. Framework for adjusting for both stress triaxiality and mesh size effect for failure of metals in shell structures. *Int J Crashworthines* 2014;19:1–12.
- [22] Kõrgesaar M, Remes H, Romanoff J. Size dependent response of large shell elements under in-plane tensile loading. *Int J Solids Struct* 2014:1–10.
- [23] Kõrgesaar M, Kujala P. Experimental validation of failure criterion for large complex shell structures. *International Conference of Collision and Grounding of Ships and Offshore Structures*, 2016 Ulsan, South Korea.
- [24] Storheim M, Alsos HS, Hopperstad OS, Amdahl J. A damage-based failure model for coarsely meshed shell structures. *Int J Impact Eng* 2015(83) pp 59–75.