2016/2500M-0058

ISSN 1797-3457 (verkkojulkaisu)
ISBN (PDF) 978-951-25-2854-7

TIIVISTELMÄRAPORTTI (SUMMARY REPORT)

Normaali- ja korkealujuusterästen törmäyskäyttäytymisen mallintaminen suurissa rakenteissa kylmissä olosuhteissa

Normal and high strength steel behaviour in low temperatures for impact prediction

Mihkel Körgesaar, Jani Romanoff*
Aalto-yliopisto, Insinööritieteiden korkeakoulu, Meritekniikka
PL 14300, Puumiehenkuja 5A, 00076, Aalto, Finland
Tel. +358 50 511 3250
Email: jani.romanoff@aalto.fi

Tiivistelmä: Laivojen yhteentörmäykset, karilleajot sekä vieraiden esineiden tunkeutuminen laivan rungon läpi johtaa laivan uppouman tilavuuden menetykseen ja mahdollisesti etenevään vuotoon laivan eri osastoissa. Syntyneen reiän koko ja paikka määrittävät vuodon etenemisen nopeuden ja laajuuden. Tämän takia reiän koon ennustaminen on ensiarvoisen tärkeää, jos mietitään yleistä laivaturvallisuutta. Suomen rannikko on matalavesinen mikä mahdollistaa karilleajot. Talvimerenkulku sekä Suomenlahden vilkas laivaliikenne lisäävät laivojen yhteentörmäysten todennäköisyyttä. Ulkoinen uhka kriisitilanteissa lisää sekä näiden tapahtumien todennäköisyyttä, että tuo mukanaan ballistisen uhan. Kylmä ilmasto laajentaa ongelmaa, koska on tiedetty, että terästen murtokäyttäytyminen muuttuu lämpötilan seurauksena. Vedessä olevan laivan rungon lämpötilagradientti voi olla merkittävä ja tätä vaikutusta pitää tutkia. Hankkeessa tutkitaan normaali- ja korkealujuusteräsrakenteiden mallintamista elementtimenetelmää hyväksikäyttäen. Tutkimuksessa keskitytään kuorielementtien toimivuuden optimoimiseen niiden laskennallisen tehokkuuden takia. Työssä tehtiin kvasi-staattiset materiaali- ja lujuuskokeet sekä simuloidaan nämä elementtimenetelmää käyttäen. Tutkimus osoittaa, että kuorielementtien käyttö on laskennallisesti edullinen ja tarkka tapa mallintaa suuria rakenteita, mikäli elementin jännitys- ja venymätila sekä koko huomioidaan määritettäessä materiaalin murtovenymää.

Abstract: Ship collisions, groundings and penetration of objects through the shell plating of the ship can lead to loss of ship buoyancy and possibility to progressive flooding inside the ship. The size and location of the opening therefore define the seriousness of the incident and form initial conditions for the prediction of the flooding process and this way define also the time to evacuation and extent of the final damage. Therefore, prediction of the opening size has fundamental importance in ship safety assessment. Finnish coast is shallow and this increases possibility for groundings. Frequent cross-traffic further increases the risk of grounding. Due to possibility for external threat the ballistic issues might become important. Cold temperature and its effects to hull girder temperature gradient makes the problem challenging as the steel used in shell plating is known to have temperature dependent stress-strain relation and also the fracture point. This project focuses on Finite Element simulations based on shell elements and their validation with material and structural testing. The study shows that use of shell elements is feasible as it can result in accurate predictions in reasonable computing time. However, this requires that the failure strain is defined based on both stress and strain state and element size instead of the commonly used scaling based purely on tensile testing.

1. Introduction

Growing awareness of environmental risks related to storage and transportation of chemicals and fossil fuels has increased the pressure to develop structures more resistant to impact and collision loads. Thin-walled structures used broadly in storage and transportation of hazardous substances inflict the highest risk for the environment, property and humans since they are especially vulnerable to puncture. In sea transportation, puncture of structures can occur due to ship collision, grounding or hostile action. The resulting opening in a side or bottom structure of an oil tanker or chemical carrier can cause serious environmental damage especially in sensitive shore areas. These accidents are potentially very costly due to clean-up operations and lost property, and thus are a burden to society. Therefore, structure's ability to resist high local load without fracturing can be the only factor preventing disaster. Therefore, understanding fracture phenomenon and ability to simulate fracture in large thin-walled structures is crucial in order to design collision-resistant or crashworthy structures. Furthermore, the concept of crashworthiness lends itself to broad class of military infrastructure that in certain situations are subject to large deformations and fracture.

Simulation based design is the most cost-effective and fastest approach to determine the response of shell structures in case of fracture. However, when the structures being analysed become large, detailed modelling of the fracture process becomes prohibitive, considering current software and hardware capabilities. Therefore, analysis of large thin-walled structures is commonly performed with large shell elements due to the limits of computational capacity. While computationally efficient, the size of the large structural shell elements imposes restrictions on how the fracture initiation and propagation can be modelled in large structures. In other words, the governing challenge is how to represent micro-scale phenomena of fracture initiation with large plane-stress shell elements, see attached figure.

In overcoming these challenges, the first step is to characterise the material behaviour until fracture under different stress state. In this context, stress state is important since material fracture ductility, i.e., how much we can deform the material before it fails, depends on the stress state. Once we know the material behaviour until fracture, we have to adjust the fracture strain to be suitable for large shell elements used in practical finite element simulations. Whether the adjustment made are reasonable can be only assessed through comparison with experimental results.

2. Research objectives and accomplishment plan

The objective of this project is to determine how well we can characterize the structural performance under large deformations using non-linear finite element simulations based on shell elements and state-of-the-art fracture modelling approaches. This is achieved through comparison of numerical and experimental results. Therefore, the specific objectives of present study are:

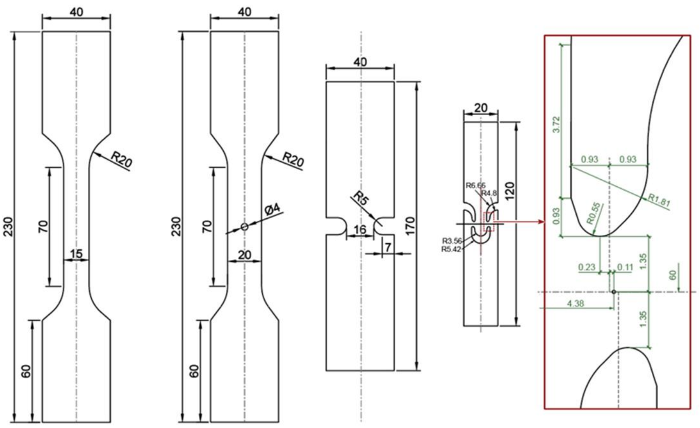
1. Determination steel behaviour until fracture initiation at different stress states

using custom defined tensile specimens in of thin-walled structures (thickness of 1.5 and 3 mm). This includes the determination of fracture curve that relates fracture strain to specific stress state. Corresponding numerical simulations are performed to validate the material curve that will be used in large-scale panel simulations.

2. Determination of structural response of sandwich and stiffened panels under large deformations using clamped plate experiments. Force and displacement are the main variables recorded during experiments and the energy absorbed by the structure is defined based on this. In some tests, also digital image correlation will be used to measure surface displacements and strains are to be extracted from these using the basic assumptions of continuum mechanics.
3. Comparison of sandwich and stiffened panel performance regarding crashworthiness and energy dissipation. This includes the validation of the simulation approach for both structures and identification of the problems in simulations. Focus in on force–displacement- relations and observed fracture paths. Also the influence of cold temperature based on measured trends is identified.

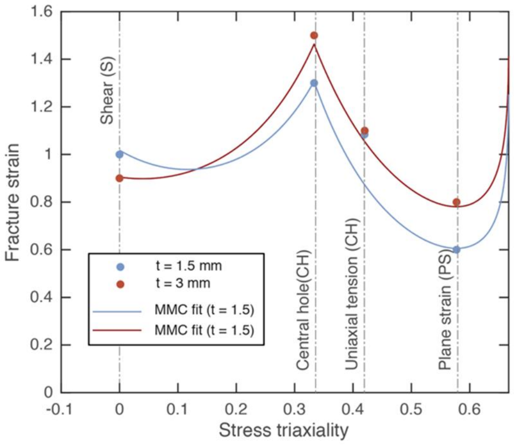
3. Materials and methods

Material of specimens is standard structural steel S235JR with following minimum values for mechanical properties defined in standard EN 10025-2: ReH = 235MPa, Rm = 360…510MPa. Two plate thicknesses are considered: t=1.5 mm and 3 mm. The thinner plates are used in face plates of steel sandwich panels, while the thicker plates elsewhere. To characterize the mechanical properties of this steel at different stress states, quasi-static tensile tests were performed at room and low temperature with different tensile specimens; for details of the experimental set-up and analysis see the report M1-Tensile Experiments in Appendix A. In total, four different type of specimens were tested to map the plane stress fracture locus in the space of equivalent plastic strain and stress triaxiality; these shapes are obtained by following the ideas from scientific literature and by iteration based on assumed material curve (in attached figure A) Uniaxial tension (UT), b) central-hole (CH), c) plane strain (PS) and d) shear (S) specimen). Specimens were custom defined and optimized to characterize material fracture ductility under different stress states. In room temperature, four tests were performed with each specimen, and in low temperature, two tests were performed with each specimen. During testing, the force and displacement were recorded. Additionally, in room temperature tests displacement fields on the one side of the specimen surface were recorded with a high-resolution digital camera. The images were post-processed with digital image correlation (DIC) software to acquire local strain data.

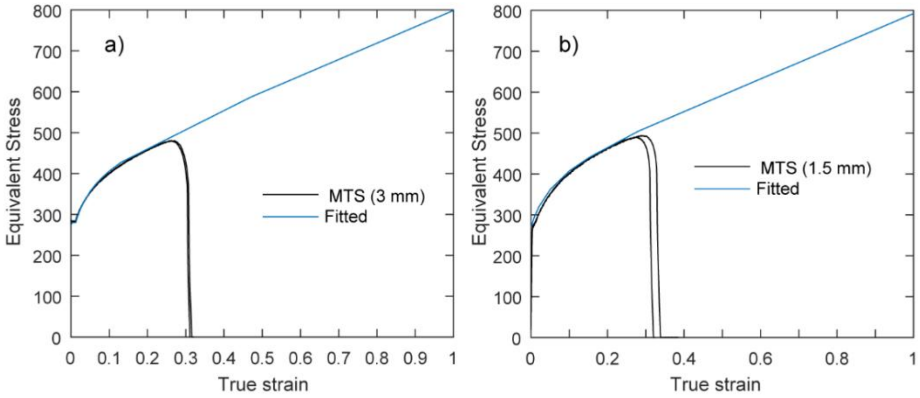


a) UT b) CH c) PS d) S

The fracture strain for each specimen was determined through hybrid experimental-numerical procedure; for details, see M1-Tensile report. This involves analysis of DIC data and corresponding numerical simulations with each test until fracture to obtain the stress and strain histories at the material point where fracture initiates. Numerical simulations play a central role in developing, analyzing and validating ductile fracture experiments. Until the point of necking true stress-strain curve or the material hardening curve is estimated from the corresponding engineering values determined with uniaxial tension (UT) specimen. An inverse approach was used to represent post-necking part of the hardening curve meaning that input true stress-strain curve was iterated until good correspondence between numerical and experimental results.



The outcome of this hybrid procedure is plotted in attached figure together with the Modified Mohr-Coulomb (MMC) plane stress fracture criterion (Bai and Wierzbicki, 2010) fitted through the measured data. Stress state dependent fracture strain obtained with different tensile specimens. To characterize the fracture strain over whole stress triaxiality range MMC fracture criterion is fitted through the data.

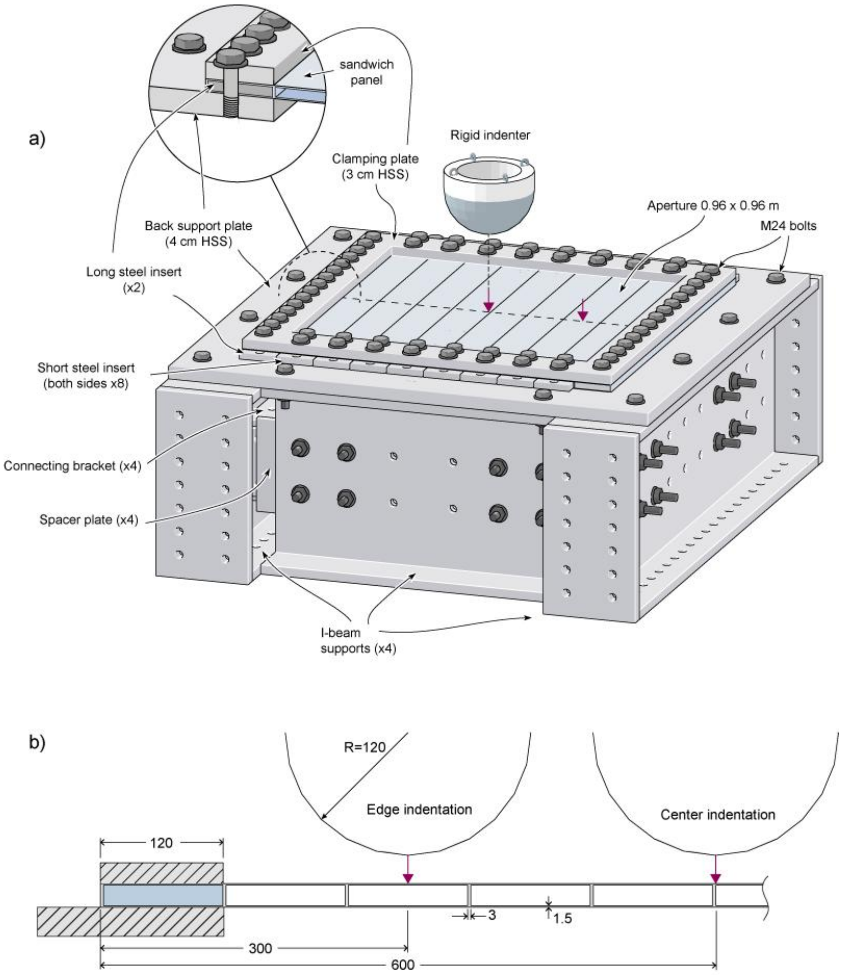


Laser welded stiffened and sandwich steel panels were indented quasi-statically with a spherical steel bulb to simulate collision of thin-walled structure, e.g. ship hull with rigid object. Experiments function as a benchmark for collision simulations. They also allow to investigate the general performance of panels under large deformations and especially their fracture process from initiation to propagation. They also reveal the behaviour of welds and their influence on the panel performance. Two types of stiffened steel plates were tested; sandwich panels (SW) and stiffened panels (SP), five pieces of both. Panels were manufactured by Koneteknologiakeskus in Turku, Finland from steel sheets produced by SSAB. Illustration of the experimental set-up is show in attached figure. The design of the support frame and clamping configuration was motivated by the large number of tests performed. The full test matrix is described in Table 1. The main output from the experiments was indentation force and displacement of the indenter. These will be compared with simulations. Details regarding the specimens, experimental set-up and measurements can be found in the M2-Panel report.

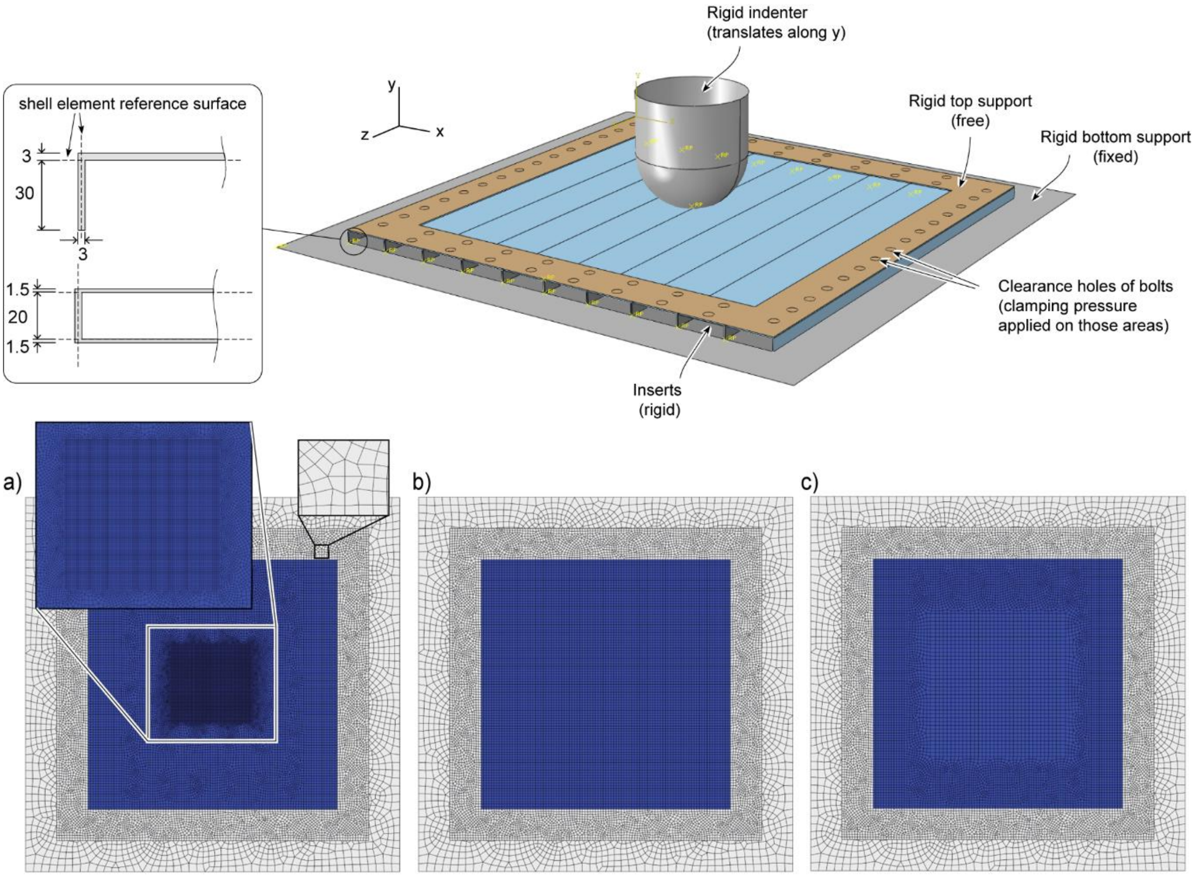
Table 1. Test matrix

Specimen ID	Hit location	Date	Offset		Notes	DIC
			dx	dy		
SP1	edge (base)	01/09/2016	618	900	F=320, indenter inclined, no fracture in the plate	no
SP2	center	31/05/2016	603	595		no
SP3	center	xx/xx/xxxx	602	593	reduced panel, including only sections 4,5,6 & 7	no
SP4	center	27/06/2016	598	593		yes
SP5	center	12/07/2016	597	592		yes
SP6	center	17/08/2016	597	593		yes
SW1	specimen cutting					
SW2	Center	30/08/2016	601	599	coordinate system numbering switched	no
SW3	edge (node)	02/09/2016	610	840	coordinate system numbering switched, 220 kN big bang and indenter	no
SW4	edge (base)	31/08/2016	610	900	coordinate system numbering switched	no
SW5	center	23/08/2016	600	595	coordinate system numbering switched	yes
SW6	center	24/08/2016	600	598	coordinate system numbering switched	yes

Finite element analyses of indentation experiments were carried out to study how boundary conditions, load-carrying mechanism and material behaviour contribute to the behaviour of panels. Thus, a series of simulations was run where different simulation parameters and modelling tech-niques were applied, for details see M2-Panel report. By comparison of experimental results and different simulation results it was possible to reveal validity of experimental results and modelling techniques. Simulations were run using Abaqus/Explicit version 6.13-3 using reduced integration shell elements (S4R) with default hourglass control and 5 through thickness integration points; see figure below. Two fracture criteria were used in simulations: one state-of-the-art in marine structures community that consider uses uniaxial tensile test results to extract the failure strain for different element lengths and the other that considers the effect of stress state on the fracture strain and other developed in Aalto University that does. The fracture criteria used in simulations are also detailed in the M2-Panel report. To investigate the element size sensitivity of the FE solution, a total of three different mesh densities were adopted in models as shown in figure below: 3 mm, 10 mm and 15 mm mesh. A 10 mm meshed model was used as a seed model for 3 mm and 15 mm models, which were thus obtained after subsequent refinement or coarsening at the indentation location, respectively. The coarse mesh in the 15 mm model was adopted only in the central region as opposed to the whole panel to keep the mesh resolution at the clamping location similar between panel and



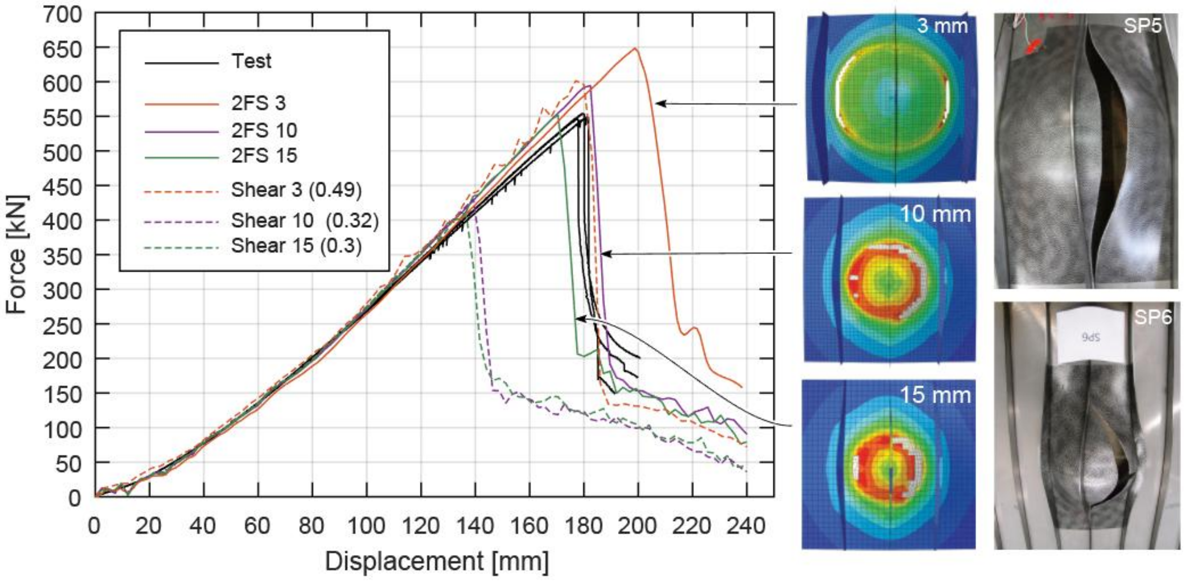
top clamping plate – the latter was meshed with seed size of 10 mm. This constraint of 10 mm was imposed by the size of the clearance hole areas that were explicitly modelled to exert clamping pressure.

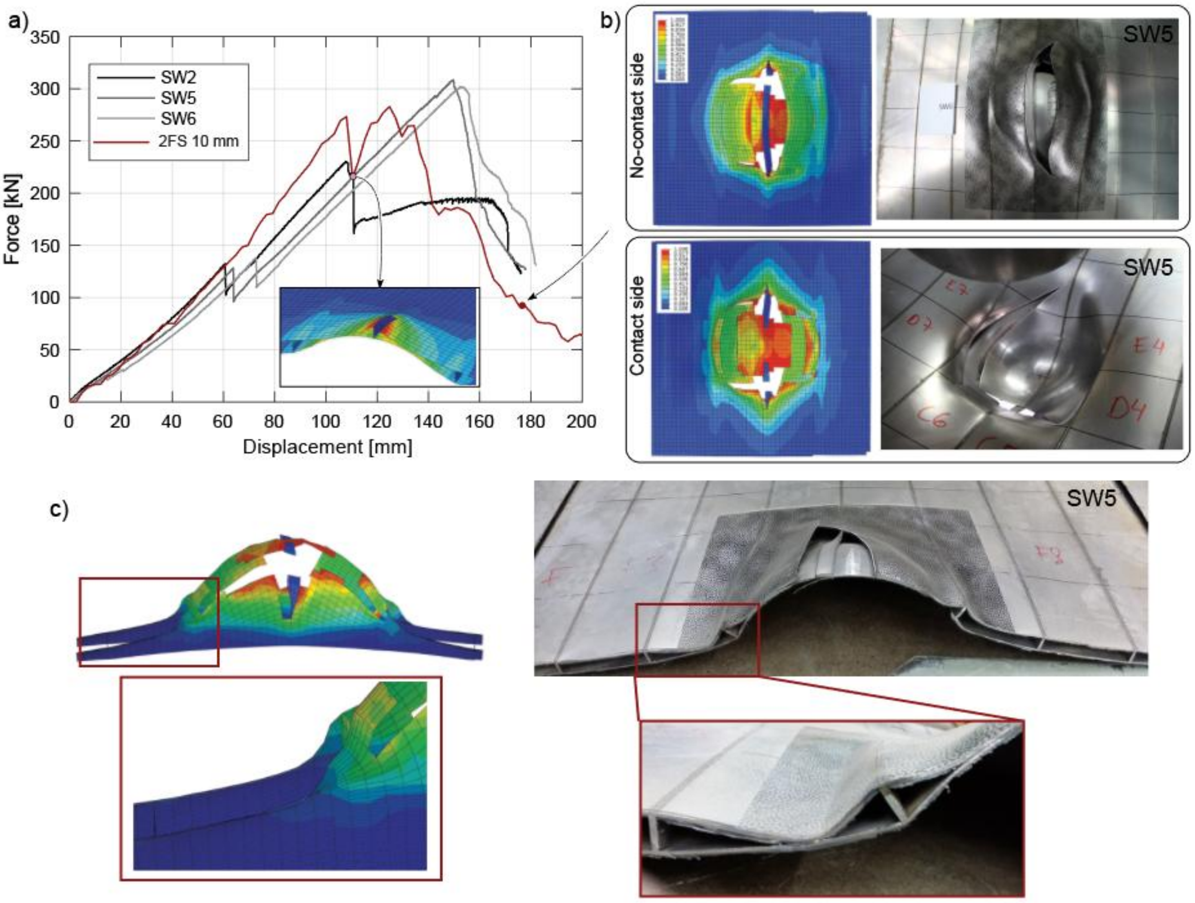


4. Results and discussion

The results of the tensile tests showed that repeatability of the tests was good with some variations in fracture strain for shear specimens. Although local strains at the specimen surface were measured using high speed cameras and Digital Image Correlation system, the post-processing of the image data was more time consuming than expected. Therefore, the fracture strain for each tensile test was determined from finite element simulations. The analyses show the comparative fracture strain determined with local strain measurements, but this will be reported elsewhere. The simulations performed showed that simple von Mises plasticity works reasonably well for all specimens, but had some difficulties capturing the shear dominated response of shear specimens. This should be kept in mind in future investigations. Measured force-elongation curves were compared with simulated curves and details are given in M1-Tensile report. The main objective of determining the fracture strain for different stress states was achieved.

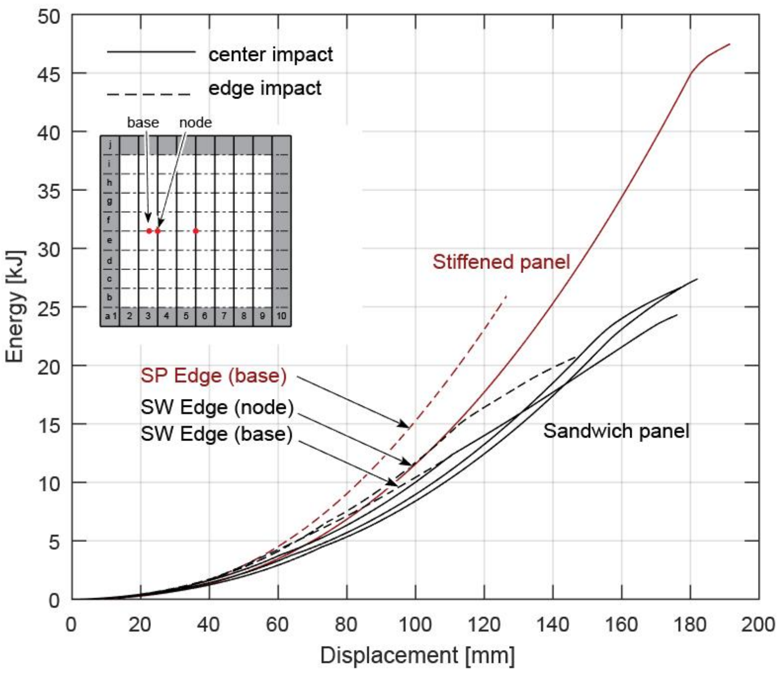
Figures below compare the simulated and experimental force-displacement-curves for stiffened and sandwich panels respectively. The state-of-the-art, uniaxial criterion (independent of stress state) provides an accurate estimate for fracture initiation when element size is 3 mm, but once the coarser mesh is used the criterion considerably underestimates the performance of the structure with fracture taking place at 140 mm. Conversely, when 2FS criterion (AALTO, stress state dependence) is used displacement to fracture is overestimated with finest mesh, but accurately captured by coarse mesh. The 2FS criterion encouragingly reduces the mesh sensitivity of the solution that is the main hallmark of the criterion.





Besides the main objective of determining the point of fracture initiation, we made an inquiry about fracture path. In the figures above the fracture path is shown for all mesh densities together with post-mortem pictures of two panels. One of the findings was that fracture always occurred on the opposite of the indentation location across the stiffener – in case of SP5 and SP6 indentation location was ~8 mm to the left of the stiffener. Both coarse mesh models capture this well, but not the 3 mm model. Furthermore, at later stages of simulations fracture propagated through the stiffener to the other side while this did not happen in experiments. This could be explained by the fact that indenter being constrained against rotations only by springs, immediately started to rotate incrementally after fracture initiation due to the associated drop in resistance, and thus unloading the opposite side. However, in simulations rotational degrees of freedom were constrained as the initial objective was to capture the point of initiation associated with the load drop. The effect of indenter rotations on the crack path will be further studied. Also the fact that in 3mm elements the plane stress assumption of shell elements is seriously violated due to formation of local necking, the case needs further investigations when shell elements are used.

The sandwich panel simulations were performed only with 10 mm mesh as the results for stiffened panel were the most accurate. The sandwich panel behaviour was less predictable than stiffened panels, due to the fact that in all panels there was a sudden drop in force at about 60 mm displacement that we associated with weld failure. In simulations this is not captured which consequently leads to overestimation of the force after 60 mm of displace-ment. The effect of weld failure and how to account this in simulations will be addressed in future studies. Fracture initiation in simulations occurs similarly to experiments on the non-contact side next to the stiffener and propagation is dominantly along the straight path. In contrast, on the contact side fracture propagates along annular path – this is also well captured by the simulation. Figure above also shows the plate buckling due to the rotating stiffener which was not captured by the simulation. This might be sensitive to indentation location that might be slightly different in simulations and experiments.



Comparison of plastic dissipation energy on in attached figure reveals that stiffened panels absorb almost two times more energy than sandwich panels. Specific energy absorption of structures calculated as energy/mass is an important characteristic of crashworthy structures, and by considering that panels are of equal weight, this measure is almost 2-times higher for stiffened panels. Although sandwich panel construction makes them much efficient carrying global pressure loads, the performance under indentation loads up to fracture is disappointing and against many of the findings from scientific literature where the weld failure is not considered. Nevertheless, these results are only applicable to tested structures. The energy dissipation in sandwich as well as stiffened structures is strongly linked to relative stiffness of the constituents. For instance, by making the webs of the sandwich thinner, structure becomes softer and more efficient in distributing loads that also delays fracture initiation. One of the important parameters is also the rotational stiffness and failure characteristics of welds. Previous discussion reveals that laser welds prematurely failed in sandwich panels which negatively affects the absorbed energy. In conclusion, there are different alternatives how to increase the crashworthiness characteristics of sandwich panels, and while such design optimization is out of the scope of present investigation, current study shows that this optimization cannot be successful without corroborating experiments.

5. Conclusions

The objective of this project was to assess the accuracy of non-linear finite element approaches used to predict fracture in large-scale shell structures. Because of the computational restrictions these simulations must be performed with shell elements. The accuracy of simulations was quantified through extensive set of experiments performed with tensile specimens as well as large-scale panel specimens. The tensile experiments considered various specimen shapes to model the effect of stress triaxiality on failure strain. Also the tests were carried out in room and -20 degree temperatures to model the effect of temperature to stress-strain curve.

Tensile tests provided the necessary information regarding material plasticity and fracture behaviour. Panel experiments were performed to validate the numerical simulations. Although the relatively simple stiffened and sandwich panels do not sample all the relevant deformation path-ways, this apparent drawback limits the number of modelling approximations, idealizations, and assumptions. Thereby, in many ways they serve as better benchmark cases as more complex alternative arrangements. On the other hand, the laser welds by themselves already provided a challenging modelling aspect, and thus were left out of the present investigation.

A systematic comparison of experiments and simulations provides an invaluable insight to the validity of numerical simulations and points to the weaknesses that should be considered in future investigations. It was shown that through systematic investigation of material properties and sub-sequent modification of fracture criterion we can have more reliable numerical simulations where element size effect is considerably reduced. Nevertheless, some calibration effort is still needed to further reduce the element size sensitivity, especially when using relatively fine mesh (element length of 3 mm). This work is currently ongoing.

In spite, the time instance and location of fracture initiation was encouragingly captured in stiffened panel simulations. The behaviour of sandwich panels was less predictable, both in tests and simulations. Experimentally observed load drop associated with laser weld failure was not captured by simulations, although fracture path on both sides of the panel was qualitatively similar to tested panels. Further insight into the behaviour was gained by cutting the sandwich panel across the mid-length. Placing the pictures of sectioned panel side-by-side with simulated panel revealed that deformations in the adjacent cells of the sandwich were different, possibly because of the rotation stiffness of the T-joint laser weld.

The comparison of energy dissipated during indentation showed that at equal weight stiffened panel is better energy absorber than all-steel web-core sandwich panel. Nevertheless, this finding is only relevant to particular panels and should not be extrapolated to any other sandwich panels as the properties of the panels are highly dependent on the topology and architecture of constituents. Though important notion is that some of the energy is lost because of the premature weld failure making the joining technology in sandwich panels an important characteristic.

The main focus in the investigation was capturing the point of fracture initiation, but propagation of fracture and analysis of that proved equally exciting. It was revealed that fracture path at later stages of simulations was inconsistent with behaviour observed in tests. This was believed to occurs because of the indenter rotations that were constrained in the simulations for simplification purposes.

In current experimental-numerical investigation lot of effort was put on maximizing the data that can possibly be extracted from experiments and thus provide relevant benchmark data for further use. Therefore, the digital image correlation (DIC) was used to measure the displacement fields. Although analyses of the “big data” generated during DIC measurements are still ongoing, and thus left out of the present report, we are very excited about the possible outcomes that such full-field comparison could provide – for instance, from general validation of numerical simulations to detailed investigation into the crack initiation and propagation.

6. Scientific publishing and other reports produced by the research project

Aalto University, School of Engineering, Reports:

Kõrgesaar, M., Romanoff, J., Palokangas, P. (2016). Penetration resistance of stiffened and web-core sandwich panels: experiments and simulations. Scientific Report.

Pekka Palokangas, “Quasi-static indentation experiments and simulations of stiffened steel panels”, Master thesis, Aalto School of Engineering, 2016.

2 journal and 2 conference papers are planned to be published during 2017.

Kõrgesaar, M., Palokangas, P., Romanoff, J., Bossyt, S. and Remes, H., “Penetration resistance of stiffened and web-core sandwich panels: experiments”, [Journal]

Kõrgesaar, M., Palokangas, P., Romanoff, J. and Remes, H., “Penetration resistance of stiffened and web-core sandwich panels: simulations”, [Journal]

Kõrgesaar, M., and Romanoff, J., “Sensitivity of simulation parameters to ductile fracture prediction in stiffened panels”, International Conference on Structural Integrity, Funchal, Madeira, Portugal, 4-7 September, Portugal.

Bossuyt, S., Kõrgesaar, M., Romanoff, J. and Shreyas, N., “Increasing the accuracy of large-scale crash simulations with digitization”, Conference and Exposition on Experimental and Applied Mechanics. Indianapolis, Indiana USA, June 12-15.