Iceberg shape characterization for damage assessment of accidental impacts with ships and offshore structures

Torstein Myhre

MASTER THESIS 2013
for
Stud. techn. Torstein Myhre

**Iceberg shape characterization for damage assessment of accidental
impacts with ships and offshore structures**
*Karakerisering av formen på isfjell for skadevurderinger av ulykkesstøt
med skip og offshorekonstruksjoner*

The Arctic waters are becoming an attractive area due to its large reservoir of oil and gas. Explo- rations in such areas will meet with harsh environmental elements, such as ice loads and low temperatures. The sea ice extent and thickness have diminished over the past few years due to global warming. This diminishing ice may provide access to new sailing routes in these waters in the years to come. The probability of collisions between ships and icebergs may increase due to this increased activity. The assessment of the loads caused by iceberg impacts is an important issue for ship designers. In modern ship and offshore structure design, the design should always be carried out under the principle of the Ultimate Limit State (ULS) and further checked with the requirements for the Accidental Limit State (ALS).

Although it is not explicitly stated, the conventional design of ship structures is carried out in the ULS format. This design implies that the structure is only allowed to undergo small deformations with no or limited elasto-plastic behaviour. The task is then to determine the scantlings such that the structure can resist the maximum pressures and forces from the ice as the ship is crushing the ice. Consequently, the ice action is characterised as pressure versus contact area curves; the smaller the contact area, the larger is the indentation pressure. Substantial efforts have been invested over the past decades to determine pressure-area relationships for crushing ice. A variety of formulations can be found in ship classification codes and rules for arctic offshore structures, such as IACS and ISO 19906. The return periods for the ice pressures are not always explicitly stated, but in general, vary from a few years to 100 years.

Pressure-area relationships are sometimes also given for very rare ice impact events, for example, for a return period of 10000 years. The intention is to use the pressure for design in the ALS format. Such pressure–area relationships are useful if the objective is to design the structure to be so strong that it can crush the ice with very moderate deformations, similar to the task in the ULS design. This approach will often lead to overly conservative design. In most codes, it is accepted that the structure may undergo substantial deformations in the ALS design; yielding, plastic mechanisms, buckling etc. are allowed, but the integrity of the structure with respect to global stability shall not be put in jeopardy. For a ship or offshore structures carrying dangerous cargo, it is also normally required that a spill into the environment should not occur. For a double-hull tanker, this requirement implies that, while gross deformation and fracture of the outer shell may be accepted, puncturing of the inner shell (cargo tank) should not occur. For this kind of design, pressure-area curves are meaningless; the structural resistance will to a large extent limit the pressures on the ice-structure interface.

Icebergs may take on a variety of shapes. A few attempts have been made to characterise the shapes, but so far, a common agreement on standard shapes to be used for ALS design has not been established. An alternative approach to characterisation of iceberg shapes on the basis of empirical surveys is to characterise the shapes in view of *the structural resistance* to iceberg impacts. In principle, a "spear-like" protrusion of an iceberg has a large puncturing potential,but in such cases the side of the ship will simply crush the ice. A vertical, plane iceberg surface yields significant confinement of the ice, which may become virtually rigid compared to the structure. However, the associated large contact area yields a significant energy dissipation capability, and no penetration of

the inner hull is likely (unless the iceberg kinetic energy is very large).

The objective of the present work is to evaluate iceberg shapes primarily in terms of their damage potential with respect to the scantlings of the shipside. As the demand for strain energy dissipation for collisions with bergy bits, icebergs and growlers up to 1000 tons is moderate, a relatively small part of the iceberg will be crushed and contribute to the energy dissipation. Consequently, it is not essential to have exact information about the overall shape of the iceberg, but instead, the local shape is important and should be evaluated in view of the structural configuration of the ship.

The work is proposed to be carried out in the following steps:

1. Describe actual iceberg shapes that have been proposed in the literature. Prepare an overview of actual ships side scantlings as a basis for the choice of parameters to be varied in a parametric study.
2. In collaboration with supervisors establish finite element models of selected shipside structures (reinforced and non-reinforced) with different dimensions and iceberg models with various shapes. The shapes shall be selected with due consideration of the side structure dimension.
3. Perform integrated simulations of iceberg impacts with the shipside. The impact direction relative to the side and other parameters (for example ice-steel friction coefficient) may be varied). The results from the simulations shall be described and discussed with respect to distribution of energy dissipation and damage in the shipside or the iceberg as well as interface forces and pressures.
4. In particular, establish pressure-area relationship of the ship side mapped on a *rigid* iceberg. The pressure area relationship for the ship side shall be compared with the pressure-area relationship for realistic ice crushed against a *rigid* ships side. Is it possible to predict the integrated behaviour from this information?
5. On the basis of the results from the numerical simulations the most "onerous" iceberg shape with respect to damage of the side structure and possible penetration of the inner shell shall be identified. Propose a characteristic shape to be used for ALIE design (the shape may depend on side structure lay-out)
6. Conclusions and recommendations for further work

Literature studies of specific topics relevant to the thesis work may be included.

The work scope may prove to be larger than initially anticipated. Subject to approval from the supervisors, topics may be deleted from the list above or reduced in extent.

In the thesis the candidate shall present his personal contribution to the resolution of problems within the scope of the thesis work.

Theories and conclusions should be based on mathematical derivations and/or logic reasoning identifying the various steps in the deduction.

The candidate should utilise the existing possibilities for obtaining relevant literature.

Thesis format
The thesis should be organised in a rational manner to give a clear exposition of results, assessments, and conclusions. The text should be brief and to the point, with a clear language. Telegraphic language should be avoided.

The thesis shall contain the following elements: A text defining the scope, preface, list of contents, summary, main body of thesis, conclusions with recommendations for further work, list

of symbols and acronyms, references and (optional) appendices. All figures, tables and equations shall be numerated.

The supervisors may require that the candidate, in an early stage of the work, present a written plan for the completion of the work. The plan should include a budget for the use of computer and laboratory resources, which will be charged to the department. Overruns shall be reported to the supervisors.

The original contribution of the candidate and material taken from other sources shall be clearly defined. Work from other sources shall be properly referenced using an acknowledged referencing system.

The report shall be submitted in two copies:
- Signed by the candidate
- The text defining the scope included
- In bound volume(s)
- Drawings and/or computer prints that cannot be bound should be organised in a separate folder.

Ownership
NTNU has according to the present rules the ownership of the thesis. Any use of the thesis has to be approved by NTNU (or external partner when this applies). The department has the right to use the thesis as if a NTNU employee carried out the work, if nothing else has been agreed in advance.

Thesis supervisors
Prof. Jørgen Amdahl
Ph.D candidates Martin Storheim and Ekaterina Kim

Deadline: June 10, 2013

Trondheim, January, 2013

Jørgen Amdahl

Preface

This thesis completes my master's degree at the Norwegian University of Science and Technology (NTNU). The work has been carried out at the Department of Marine Technology during the spring semester of 2013 under the supervision of Professor Jørgen Amdahl and Ph.D candidates Martin Storheim and Ekaterina Kim.

I would like to thank my supervisors for the guidance and motivating discussions during the work on the thesis.

Torstein Myhre
June 10, 2013
Trondheim

Abstract

The northern areas are becoming more and more attractive because off the large oil and gas reservoir that is there. In this area there is a harsh environment and the possibility to collide with an iceberg increases as the number of vessels in the area increases. The project is a master thesis that was trying to find appropriate shapes of the contact area of an iceberg in relation with the ship scantlings that can be used for design against iceberg impacts. This thesis investigated an accident with a return period of 10 000 years, which correspond to an ALS (accidental limit state) type of event. This type of event allows for some plastic deformation in the structure as long as it does not danger human life, have significant pollution or financial consequences.

By dividing the calculation in two, first by calculation of the energy in the iceberg relative to the ship and then impose the energy in a collision with the ship. To simulate the collision between the ship and the iceberg there were used non-linear finite element method by using the computer code LS-DYNA 971.

The ship structure and iceberg were modeled in PATRAN with shell elements for the ship and solid elements for the iceberg. The icebergs were modeled as a cone with a radius in front where the angle of the cone and radius of the front were varied. The ice material was modeled with a material model that was developed by an earlier Phd. student at NTNU. This corresponds to a shared energy design of the ship according the NORSOK codes three categories. The three categories are ductile design, shared energy design and strength design. The shipside scantlings was also varied with a ship without ice strengthening, a 1C ice strengthened ship and a 1A ice strengthened ship. The ship structure was limited to only a section of the shipside modeled with an elasto-plastic material model. In the analysis the iceberg was pushed into the shipside with a constant speed and the forces, deformations and pressures were measured.

The result was showing a tendency of that an iceberg with the largest radius in front equal to half of the web frame spacing and a cone angle higher than about 50° was the most onerous for the shipside, but were not tested any icebergs with larger radius.

Table of Contents

List of figures

List of tables

Notation

A	Nominal contact area
a_i	Icebergs added mass
a_s	Ships added mass
a_0, a_1, a_2	Constant from Tsai-Wu yield surface
C	Constant in pressure – area curve
D	Damage in RTCL criteria
$\dot{D}$	Damage rate in RTCL criteria
E	Young's modulus
E_{Strain}	Strain energy
F	Constant exponent in pressure – area curve
i	Step number
J_2	Second invariant of the deviatoric stress tensor
K	Strength index of power law expression
l_e	Characteristic length of the element
m_s	Mass of the ship
m_i	Mass of the iceberg
n	Power law exponent
p	Hydrostatic pressure
p_2	Larger root of the yield function

p_c	Critical hydrostatic pressure	
p_{av}	Average pressure used in pressure – area curve	
p_L	Local ice pressure	
q	Von-Mises stress	
r	Displacement	
$\dot{r}$	Velocity	
$\ddot{r}$	Acceleration	
t	Time	
Δt	Time of one time step	
$t_{element}$	Thickness of an element	
T	Expression for the tri-axial stress state	
v_s	Speed of the ship	
v_i	Speed of the iceberg	
V_n	Volume of the neck in an element	
V_r	Volume of one element except the volume of the neck	
τ	Octahedral shear stress	
τ_{max}	Critical octahedral shear stress	
η	Center of the elliptical failure criteria in octahedral shear direction	
λ	Center of the elliptical failure criteria in hydrostatic pressure direction	
$\dot{\varepsilon}$	Strain rate	

ε_f	Failure strain
ε_0	Initial failure strain
ε_{eq}^p	Equivalent plastic strain
ε_{ij}^p	Plastic strain components
ε_p	Plastic strain
ε_n	Critical strain when the thickness equals the characteristic length of the element (necking area)
ε_r	Average strain of one element except the necking area
$\dot{\varepsilon}_1$	Major principal strain rate
$\dot{\varepsilon}_2$	Minor principal strain rate
ε_{cr}	Critical strain an element
ζ	Temperature relation of maximum octahedral shear stress
σ	Stress
σ_Y	Yield strength
σ_1	Major principal stress
$\sigma_{1,cr}$	Critical major principal stress
β	Degree of biaxial straining

1. <u>Introduction</u>

The possibility of an iceberg impact with a ship structure is growing. The cause of this is the increased activity in the northern areas of the world. There are two main reasons for this, the oil and gas exploration in the northern areas are intensified. The second reason is that the ice in the North Pole is melting and thereby opening for new shipping routes.

The increased possibility for collision gives the basis for further research on ice impact with structures. The regulations for an offshore impact are for two different events either for a 100 year return period, ULS (Ultimate Limit State), or for a 10 000 year return period, ALS (Accidental Limit State), which both shall be checked.

This project is a Master thesis trying to find appropriate shapes of the contact area of an iceberg in relation with the ship scantlings that can be used for ALS design against iceberg impacts.

1.1   Code requirements

According to NORSOK [1] the ULS check should be performed with the worst of these load factor combinations.

Action combination	Permanent actions	Variable actions	Environmental action
a	1.3	1.3	0.7
b	1.0	1.0	1.3

Table 1: Action factor for ULS. [1]

If the event is an ALS type of event the structure shall not have a complete loss of integrity. This implies that some damage to the structure is allowed if the structure does not danger human life, have significant pollution or financial consequences.

According to NORSOK [2] the ALS analysis is done in two steps. First resistance to accidental actions then if the resistance is reduced it shall be checked for ULS events. The analysis can be done either by directly calculating of the effects imposed by the actions on the structure,

or indirectly by using the recommendation from the rules without further calculations. If non-linear dynamic finite element analysis is used it shall be verified that all needed effects and local failure modes are accounted for.

The regulation for the environmental action can be found in NORSOK N-003 [3] where the different loading shall have the return period given by the yearly possibility in the table below.

Limit state	Wind	Waves	Current	Ice	Snow	Earthquake	Sea level[a]
Ultimate limit state	10^{-2}	10^{-2}	10^{-1}	-	-	-	10^{-2}
	10^{-1}	10^{-1}	10^{-2}	-	-	-	10^{-2}
	10^{-1}	10^{-1}	10^{-1}	10^{-2}	-	-	m
	-	-	-	-	10^{-2}	-	m
	-	-	-	-	-	10^{-2}	m
Accidental limit state	10^{-4}	10^{-2}	10^{-1}	-	-	-	m*
	10^{-2}	10^{-4}	10^{-1}	-	-	-	m*
	10^{-1}	10^{-1}	10^{-4}	-	-	-	m*
	-	-	-	10^{-4}	-	-	m
	-	-	-	-	-	10^{-4}	m

[a] m – mean water level
 m* – mean water level, including the effect of possible storm surge
 Seismic response analysis should be carried out for the most critical water level.

Table 2: Combination of environmental actions with expected mean values and annual probability of exceedance 10^{-2} and 10^{-4} [3]

As we can see in the chase of ice loading, the ice load in ULS design should be considered with 100 year return period and with 10 year maximum wind, waves and current with the mean water level. In an ALS event the ice loading can be considered alone with a return period of 10 000 years and a mean water level.
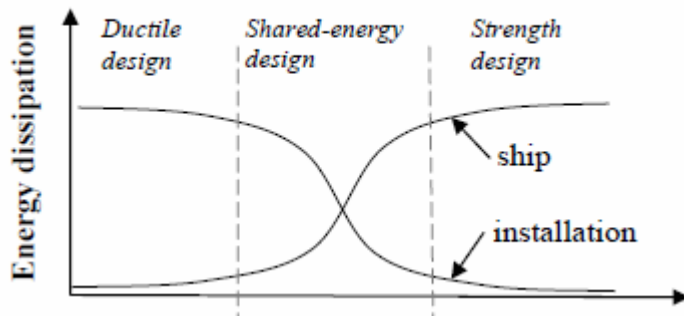
Figure 1: Relative strength [2]

For ship impacts the effects can be calculated direct or by energy considerations combined with elastic-plastic method. The structure can be designed with a ductile, shared energy or strength design as seen in Figure 1. In the strength design the iceberg would be the only body deforming. This would demand the ship to be strong enough to crush the ice. For the ductile design only the ship deforms and will therefore have to take the entire energy of the collision. This simplifies the calculations since the iceberg can be modeled as rigid. In the shared energy design both the ship and the iceberg will deform and will influence each other.

The sharing of the energy can be represented as a load-deformation curve for both the ship and the iceberg. The areas below the two curves up to a certain load level give the energy of the impact. By use of this information the impact force can be found by the graph as shown in Figure 2.
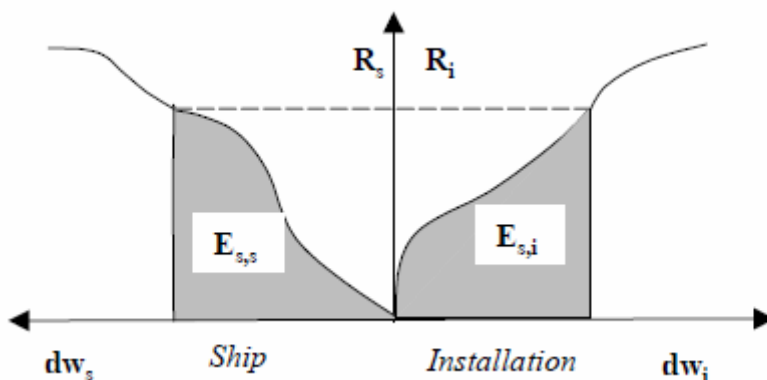
Figure 2: Dissipation of strain energy in ship and platform [2]

The collision mechanics is often divided into two methods. If the periods governing the motions are long compared to the duration of the collision, there should first be calculated the distribution of the impact energy between kinetic rotation, translation energy and deformation energy. This can be done with momentum and energy considerations, as done by Liu and Amdahl [4].

$$E_{Strain} = \frac{1}{2}\left(m_s + a_s\right)v_s^{2}\frac{\left(1 - \dfrac{v_i}{v_s}\right)^{2}}{1 + \dfrac{m_s + a_s}{m_i + a_i}}$$

(1.1)

E_{Strain} : Strain energy
m_s : Mass of ship
m_i : Mass of iceberg
a_s : Added mass ship
a_i : Added mass iceberg
v_s : Speed of ship
v_i : Speed of iceberg

The deformation energy from these calculations is then divided between the two structures. This is shown by a graph like in Figure 2. The force on both structures will always be the same. When the force is increased until the area below the graph has reached the total deformation energy, the value on the deformation axis can be read to find the deformation for the iceberg and the structure. If the duration of the collision is long compared to the Eigen period of the motion, this can be found with quasi-static method of analysis. If the period is close there should be performed a dynamic structural analysis. [3]

In the iso standard [5] there is used a pressure area curve to design against iceberg impacts. The curve is used to find the pressure to add to a load patch on the shipside.
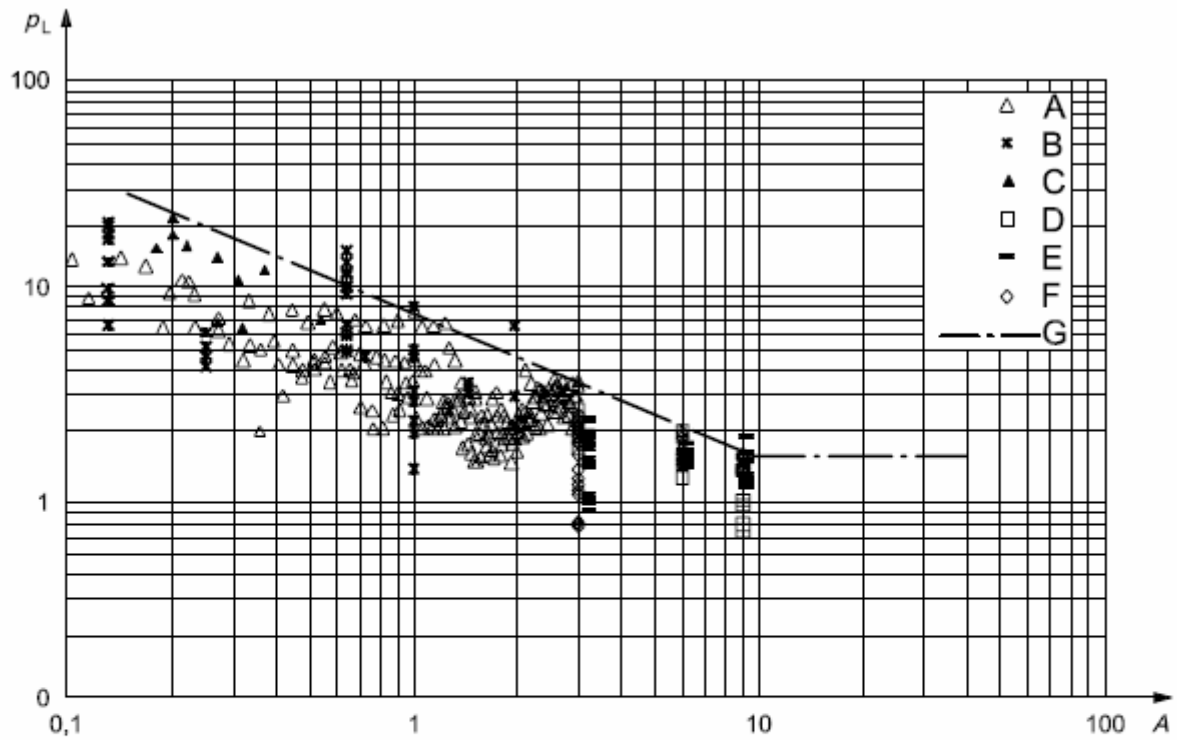
Key

A 3 m^2 Pond Inlet test data
B flat jack test data
C 1989 Hobson's Choice test data
D Molikpaq BW data
E Molikpaq N face data
F Molikpaq E face data
G $p_L = 7{,}40A^{-0{,}70}$ (mean plus three times the standard deviation), $A \leqslant 10$ m^2; $p_L = 1{,}48$, $A > 10$ m^2
p_L local ice pressure, expressed in megapascals
A contact area, expressed in square metres

Figure 3: Pressure in mega Pascal versus area in square meters for iceberg impact [5]

1.2 Nonlinear finite element method

This analysis had to be performed in a dynamic non-linear finite element program which in this case was LS-DYNA. LS-DYNA is an explicit finite element solver. In the book by Torgeir Moan [6] the solution method for a finite element solver is described. An explicit solver finds the displacement at a new time step by use of the displacement, velocity and accelerations of the previous time step. This is done by a Taylor series expansion of the displacement where the terms higher than second order are neglected.

$$r_{i+1} = r_i + \Delta t \cdot \dot{r}_i + \frac{\Delta t^2}{2} \cdot \ddot{r}_i$$

(1.2)

In this way there is no need to invert the system stiffness matrix for each step. This does the erosion technique effective to simulate cracks in the structure. This method deletes an element if the given conditions are met, such as to high strain level. A problem with deleting elements is that the mass and energy of the element also is deleted. Another way is by using double nodes, so that the elements can be split from each other by separating the nodes. This method does not delete the element and thereby is preserving the mass and energy stored in the element, but the time to solve is increased because of the greater number of nodes.

In order to keep the simulations stable the time steps needs to be very small, smaller than $2/\omega_{max}$. Where ω_{max} is the highest Eigen frequency the element model can describe. This causes high order element interpolation functions to reduce the time step and thereby increasing solving time. Therefore there are often used four noded shell elements with reduced integration so that the deflection over one element is constant and high Eigen frequencies are avoided. A simplified way of describing the time of one time step is by the time the sound uses to travel through the elements characteristic length.

There are also implicit solvers that solve the equation of motion for each time step. Thereby it needs much more memory and time to solve each time step, but can have much longer time steps. Some are even unconditionally stable which means the time step is only governed by the data wanted from the model.

Compared to linear finite element analysis, there are some considerations that have to be taken into account when doing a non-linear analysis. The material properties of steel after yield are non-linear which means the stiffness will change with the strain, and eventually be zero when the material breaks. The deflection could be larger than in a linear case were the deflections are assumed to be negligible. Therefore the load in a static case has to be applied stepwise and the stiffness relation has to be updated and iterated to reach equilibrium for each step. For a dynamic case the time is introduced stepwise. Boundary condition may also change with deflection when element contact each other. If the structure shall model buckling well it is also necessary to include initial deformation according to the tolerances for the structure not to overestimate the buckling load.

For this case, the deflection will become large when the iceberg pushes on the shipside and the initial deformations will become unimportant. The steel will then get plastic strains and eventually be large enough so that the material will break. This is simulated with deleting the elements with to high strain. When the iceberg contacts the shipside the boundary conditions for the ship side change in the contact area, and must move together with the iceberg.

1.3 Earlier studies

In earlier studies there have been used at least three different approaches to simulate the collision event. There was used the whole iceberg, just the contact area of the iceberg or just a patch load. All these tree approaches are based on experiments to try to get the typical ice material behaviour.

Gagnon [7] simulated the whole collision event with water, air, iceberg and ship, to simulate a model test and compare the result. He used a box shaped iceberg with rounded corners that was simulated with a foam material that had been adapted to ice by earlier experiments. The iceberg was at rest and the ship pushed through the water into the iceberg. The analysis gave god approximations for the peak pressure, duration and pressure distribution in the impact compared to the model test.

In a later study by Gagnon [8] the iceberg had the same shape except for the contact area between the ship and the iceberg. In this area on the model there had been added a half sphere, to simulate a pointier iceberg. The analysis gave realistic deformations compared to known accidents. The result also showed increasing pressures as the area increased.

Some analyses have been done at Norwegian University of Technology and Science by doctoral student Zhenhui Liu. Zhenhui Liu has with assistance from Jørgen Amdahl done several analyses of ice collisions. [4] In one of these analyses Zhenhui Liu developed a new formulation for the impact mechanics to account for a 3D case with 6 DOF to find the energy of the impact. He also developed a new material model for ice as described in ice material model chapter 2.1.2 [9] . He tested the ice model in an analysis with collision with the fore ship [10] and in a collision with the shipside [11]. In these studies there have been used spheres, spheres connected to a cone and parabolic shapes to simulate the icebergs local shape. The result showed that the ice strengthened ship could withstand the forces from the iceberg, but when reducing the yield strength to 75 MPa the shipside ruptured.

 In another analysis done at Norwegian University of Technology and Science by doctoral students Storheim and Kim [12] with assistance from Amdahl and Ehlers there was used five different iceberg shapes and two different material models. The material model used were the foam model Gagnon [7] developed and the material model Zhenhui Liu [9] developed.

There has been tested with the corner of a cube, a blunt ellipsoid, a sharp ellipsoid, sphere and a vertical cone. The result from the analysis showed that the material model developed by Zhenhui Liu [9] gave a softer collision because of the eroding of ice elements. The material developed by Gagnon [7] became harder and harder as the elements was not eroded away.

2 Method

2.1 Icebergs

2.1.1 The ice material
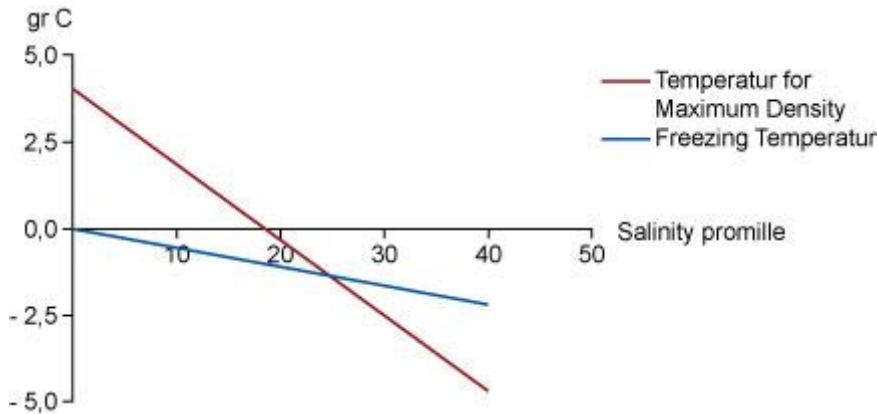
Figure 4: Freeze temperature and maximum density for water [13]

In this analysis the properties of ice were very important. According to Whitworth and Petrenko [14] the regular salt water contains about 35 parts per thousand by weight of salt and freezes at about $-2^o C$. Unlike fresh water it does not have a maximum density at a temperature above the freezing point. This causes the water on top to go to the bottom when the salt water gets colder and thereby the whole water column must be cooled down before the salt water freezes. In fresh water the convection stops when the water reaches $4^o C$ and then the ice can begin to form.
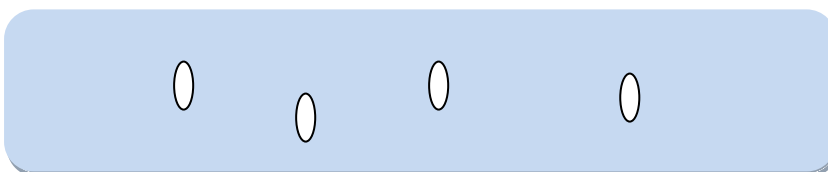
Figure 5: Brines cells in an iceberg

When the first ice forms in salt water the crystals are fresh water ice and as the ice continue to form the salt water will concentrate in brine cells inside the ice (Figure 5) and the ice will be pure except for the brine cells. As the ice gets older the salt concentration will be smaller

10

mainly due to the brines melting their way out of the ice. The ice will get stronger as a result of less brine in the ice. The new ice can also become stronger in colder temperatures when more of the brine volume freeze, and will be as strong as multi-year ice at about -20°C [5].

According to Whitworth and Petrenko [14] the ice has a different behavior for different strain rates and temperature. For compression and very low strain rates e.g. $10^{-7}[-/s]$ for $> -20^o C$, the ice will deform plastically without making cracks and will not break. For a higher strain rate, but below about $10^{-3}[-/s]$ the ice will have an elastic behavior up to a curtain load level and then deform plastically. The plastic deformation will cause cracks that will eventually break the ice. For the most of a collision event, the ice will have larger strain rate than $10^{-3}[-/s]$ and with this strain rate the ice will behave brittle. This means that the ice will have elastic strain up to a sudden fracture.

For tension the transition from ductile to brittle happens at a lower strain rate $\sim 10^{-6}[-/s]$. This is because the cracks in the ice is not pushed together as in the compressive case but are pulled open, and the failure can happen with the first crack.
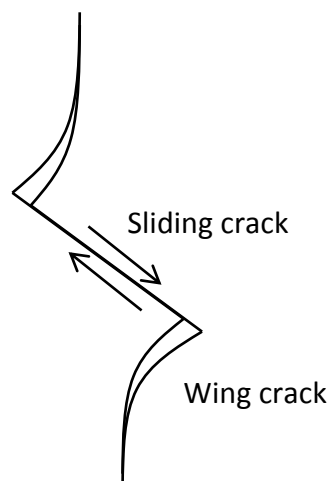
Figure 6: Wing crack

The strength of the ice is temperature dependent, and for lower temperature the ice gets stronger. For the ice to fail it has to make cracks, and under compression these cracks surfaces have to slide against each other and make wing cracks Figure 6. When the

temperature become lower the friction between the ice surfaces become higher and the force needed to slide therefore becomes greater. The brittle failure happens because of the cracks reaches all the way through the ice. At lower temperatures the crack need more force to expand and therefore the ice becomes stronger.

Icebergs are ice that is broken of large glaciers that are close to open water. The glaciers are created over many years by snowing and are therefore fresh water ice. In [5] there is given formulas for different mechanical properties of ice depending on temperature, salinity and density. Typical values for the Young's modulus of pure ice are 9-10 GPa [5] and 4 – 12 MPa for compressive strengths depending on strain rate [5].

The tensile strength in the region above the ductile to brittle transition is different for vertical and horizontal direction in sea ice due to it being columnar and is presented in Table 3.

Temperature ^{o}C	Ice type	Salinity $^{o}/_{oo}$	Action direction	
			Vertical [MPa]	Horizontal [MPa]
-10	Sea ice	4-6	1,7	0,9
-10	Baltic ice	0,2	2,2	1,0

Table 3: Tensile strength of ice [5]

2.1.2 The ice material model

The ice material model for the iceberg developed by Liu, Amdahl and Løset [9] is based on the material model developed by Derradji Aouat [15] and tested in LS-DYNA. The ice were assumed to behave linear elastic up to the yield surface than perfectly plastic until failure to simulate ice higher strain rate than $10^{-3}\left[-/s\right]$. In an impact there will be high strain rates so it is a good approximation to use a linear elastic perfectly plastic material model. The 'Tsai-Wu' yield surface in $p-J_2$ space was used as the yield surface.

$$f\left(J_2,p\right)=J_2-\left(a_0+a_1p+a_2p^2\right)=0 \tag{2.1}$$

Where J_2 is second invariant of deviatoric stress tensor, a_0, a_1 and a_2 is constants and p is the hydrostatic pressure. The surface was adapted from Derradji Aouat [15] elliptical failure criteria where η is set to 0.

$$\left(\frac{\tau-\eta}{\tau_{\max}}\right)^2+\left(\frac{p-\lambda}{p_c}\right)=1 \tag{2.2}$$

Where η and λ is the center of the ellipse, and $\tau_{\max}$ and p_c is the maximum values on the octahedral shear stress axis and hydrostatic stress axis. The adaption between the 'Tsai-Wu' yield surface and the elliptical failure criteria can be found by the following relation.

$$a_0=\frac{3\tau_{\max}^2}{2p_c^2}\left(-\lambda^2+p_c^2\right) \tag{2.3}$$

$$a_1=\frac{3\tau_{\max}^2}{2p_c^2}\lambda \tag{2.4}$$

$$a_2=-\frac{3\tau_{\max}^2}{2p_c^2} \tag{2.5}$$

$$q = \sqrt{3J_2}$$ (2.6)

q is the Von Mises stress, τ_{max}, λ and p_c are found by fitting the yield surface to experimental data. Since the material properties of ice is dependent on the ice temperature

There were also developed an expression for the cut off pressure used in the erosion function in LS-DYNA based on trial and error.

$$\varepsilon_f = \varepsilon_0 + \left(\frac{p}{p_2} - 0.5 \right)^2$$ (2.7)

p is the hydrostatic pressure, p_2 is the larger root of the yield function and ε_0 is the initial failure strain. The erosion is a function that deletes an element if the given conditions are met. In this chase, if $\varepsilon_{eq}^p > \varepsilon_f$ or if the hydrostatic pressure reach a cut of pressure.

$$\varepsilon_{eq}^p = \sqrt{\frac{2}{3} \varepsilon_{ij}^p : \varepsilon_{ij}^p}$$ (2.8)

Where ε_{eq}^p is the equivalent plastic strain and ε_{ij}^p is the plastic strain components.

The following parameters were used for the ice material

Density	Young's modulus	Constant	Constant	Constant	Tensile cutoff	Initial failure strain
ρ	E	a_0	a_1	a_2	ε_f	ε_0
$900 kg/m^3$	9.5 GPa	22.93 MPa^2	2.06 MPa	-0.023	-2 MPa	0.01

Table 4: Ice material properties

2.1.3 Chosen iceberg shape
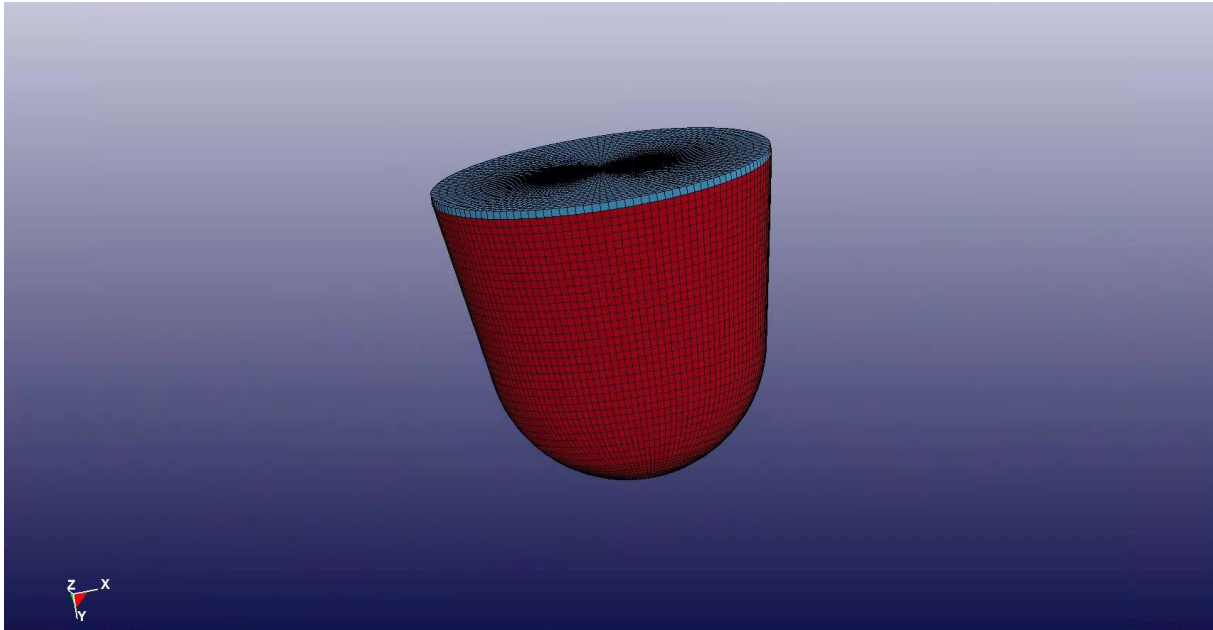
Figure 7: Iceberg shape

The shipside's weakest point will be midway between the frames with respect to getting the largest deflection and the least steel to penetrate to reach through the shipside. The sharper the iceberg is the more likely is rupture of the plating on the shipside and the further in the iceberg will reach with a given energy. Therefore the midpoint position will be tested in this study with several dimensions of the iceberg to see the sharpest shape which the iceberg can have without being crushed by the shipside. Since the ice is likely to be crushed in a sphere shape in the contact area this analysis will be performed with an iceberg formed as a cone with a radius in the front. This way it was easy to change the radius to get a sharper or blunter iceberg.
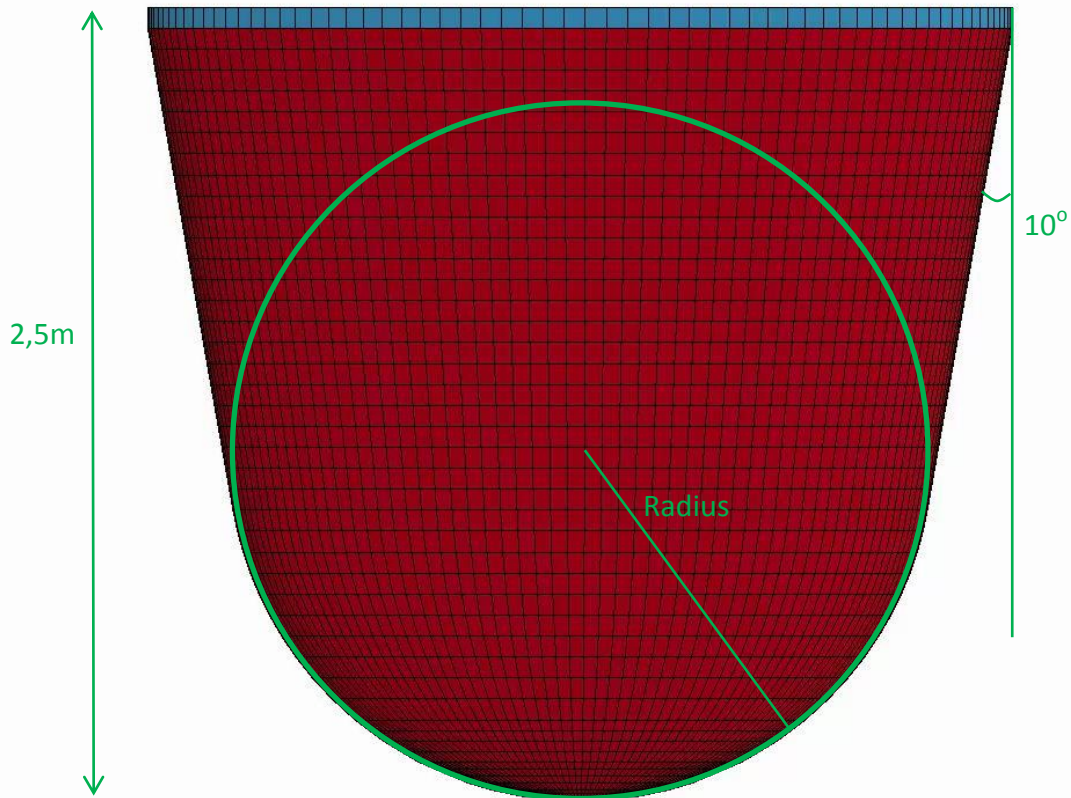
Figure 8: Iceberg dimensions

The values chosen for the iceberg is as shown Figure 8, and the radius were varied. The effect of confinement was also considered with change of the cones angle. The value for the length was chosen to be 2.5m to get a good length of ice left to the boundary after having run 1 m of the iceberg into the ship.

2.1.4 Mesh density

To mesh the iceberg it had to be divided into slices where each slice was meshed with the iso mesh function in Patran. Since the ice material model was designed to be used on hex mesh, the center of each disk caused problems because of the surface of the elements facing the center would be of zero length. This was solved by cutting away the center of all the disks and then making a surface in the center of one of the disks. This surface was meshed using the paver function in Patran which makes an unstructured mesh to avoid the zero length problems with the iso mesh. Then extrude the mesh through the center of all the disks to make solid hex elements.

The mesh size was checked by a sensitivity study where the mesh size at the flat end of the iceberg was adjusted between 100mm and 400mm cube shaped elements. The angle of the cone and radius in front caused the elements to become smaller at the radius end. Here the side of the elements will be approximately $\frac{1}{10}, \frac{1}{20}$ and $\frac{1}{3}$ of the size in the other end.
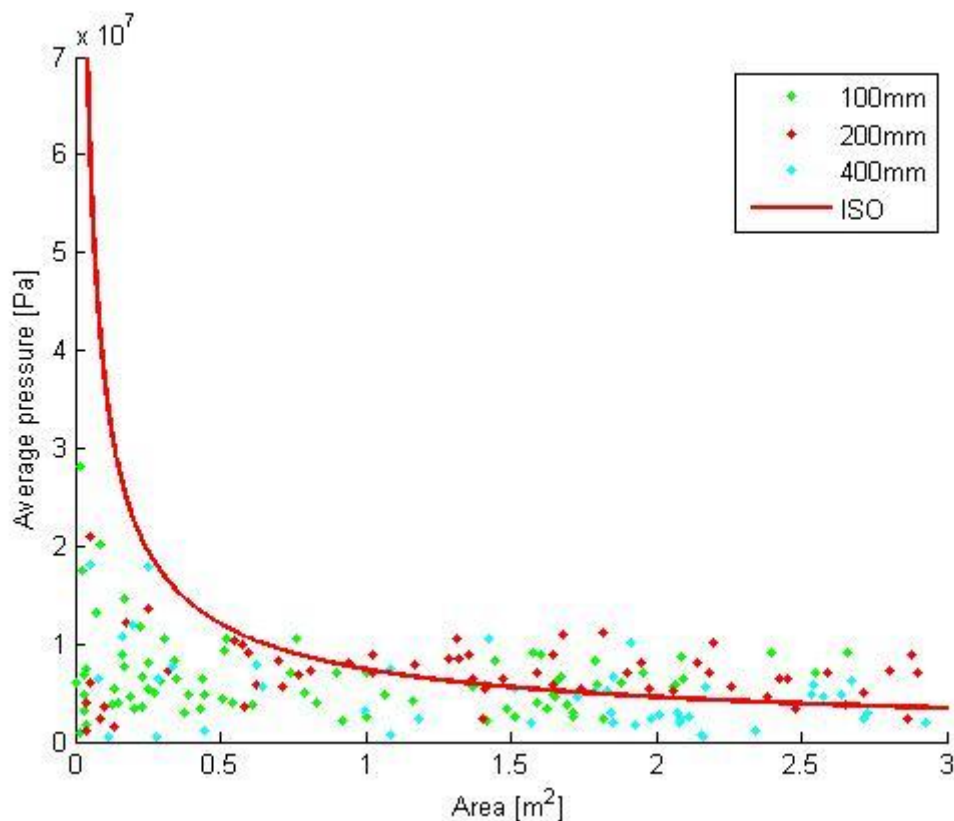
Figure 9: Average pressure-area graph for mesh size check of the iceberg. The legend gives the size of the elements on the side of the iceberg that is away from the ship and will be smaller near the ship.
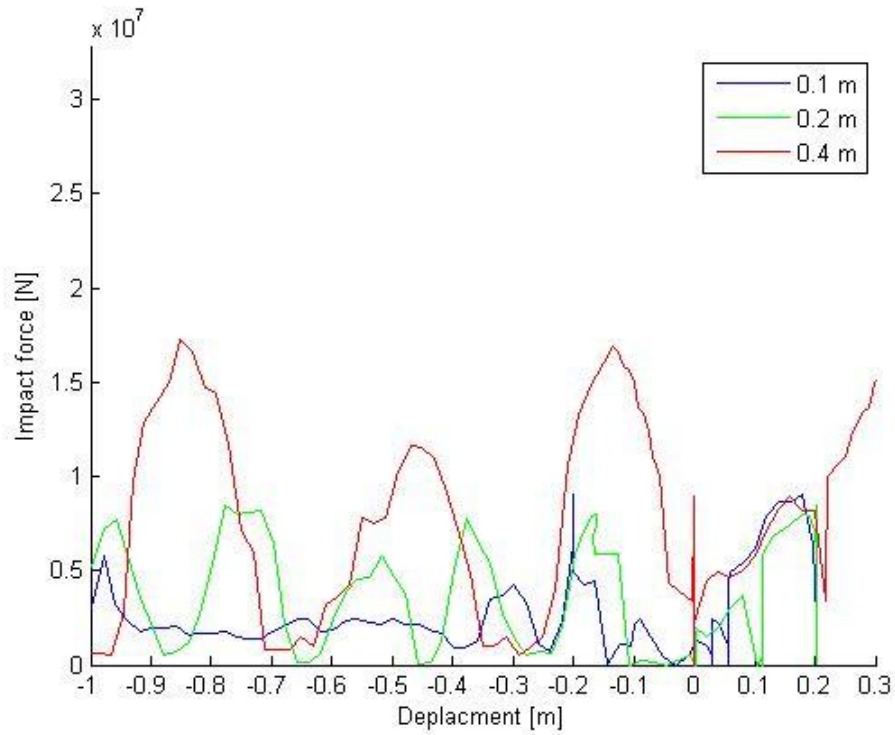
Figure 10: Impact force-displacement graph for mesh size check of the iceberg. The legend gives the size of the elements on the side of the iceberg that is away from the ship and will be smaller near the ship.
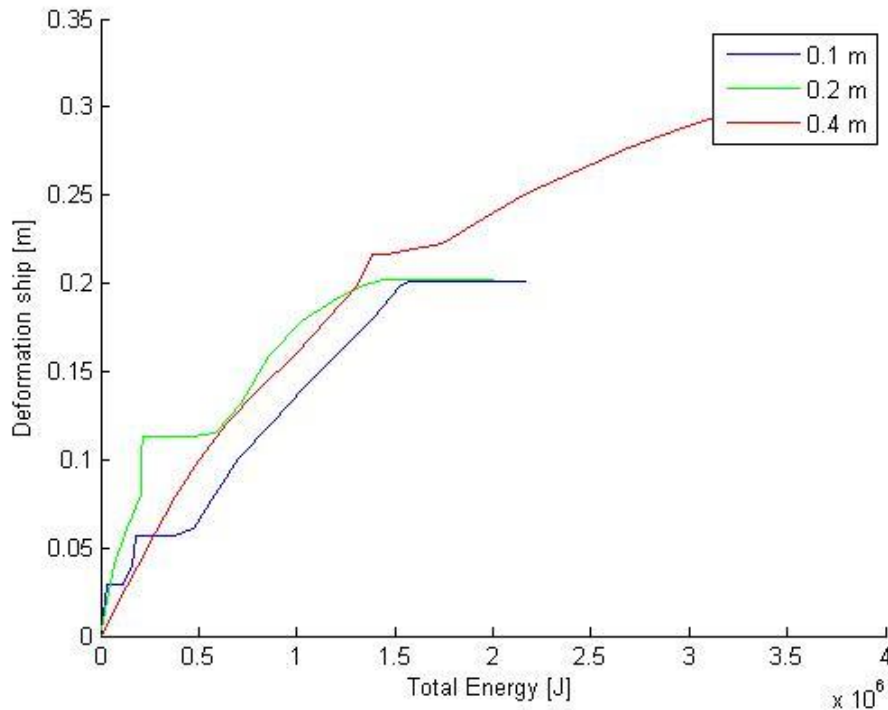
Figure 11: Deformation total collision energy graph for mesh size check of the iceberg. The legend gives the size of the elements on the side of the iceberg that is away from the ship and will be smaller near the ship.

Figure 9 shows little difference in pressures between the different meshes. The difference is seen in the impact force-displacement graph in Figure 10 which shows that the impact force become smother with smaller elements. The smallest elements give a force around the average of the other meshes, and a deformation of the shipside equal to the second smallest elements. The two smallest elements give about the same displacement, energy and pressure. The smallest elements were chosen of these two to better describe the shape of the iceberg as well as giving smother impact forces.

2.2 Shipside

2.2.1 Steel material model

The material used for the steel was based on power law hardening rule with RTCL damage.

The following values were used for steel material.

Density	Young's modulus	Yield strength	Strength index	Strain hardening exponent	Strain at failure	Friction
ρ	E	σ_Y	K	n	ε_n	
$7890 \; kg/m^3$	210 GPa	285 MPa	740 MPa	0.24	0.7	0.15(Corroded steel table A 8-5 in [5])

Table 5: Steel parameters

The material model uses the power law hardening rule, which defines the curve after yielding like the following equation.

$$\sigma = K\varepsilon_p^n \tag{2.9}$$

Where K is the strength index, ε_p is the plastic strain and n is the strain hardening exponent.

The shell element is sensitive to element size when the deformation becomes large. According to Alsos [16], when the mesh is fine the local necking of the plate will cover a larger area of the elements so that the strain in the element need to account for the thinning of the necking zone. This was done by assuming plastic incompressibility and establishing the following relationship.

$$\varepsilon_{cr} = \frac{\varepsilon_r V_r + \varepsilon_n V_n}{V_{el}} \tag{2.10}$$

$$V_n = \left(t_{element}\right)^2 \cdot l_e \tag{2.11}$$

$$V_r = V_{el} - V_n \tag{2.12}$$

Where ε_n is the average strain in the neck, ε_r is the average strain of the rest of the element. $t_{element}$ is the thickness of the element, l_e is the characteristic length of the element. V_{el} , V_n and V_r is the volume of the element, the neck and the rest of the element, which is constant due to plastic incompressibility. By using that the neck has a width equal to the thickness of the element there were found this relation between element size and critical strain.

$$\varepsilon_{cr} = n + (\varepsilon_n - n)\frac{t_{element}}{l_e}$$

(2.13)

Where n is the strain hardening exponent, $t_{element}$ is the element thickness, l_e is the characteristic length of the element, ε_{cr} is the critical strain, ε_n is the critical strain when the thickness equals the characteristic length. The value for the critical strain can then be introduced to the RTCL criteria.

The RTCL damage describes the damage to one element by using an evolution rule. [16]

$$T = \frac{p}{q}$$

(2.14)

$$\dot{D} = \begin{cases} 0 & if & T < -\frac{1}{3} \\ \frac{\sigma_1}{q}\varepsilon_{eq} & if & -\frac{1}{3} \le T < \frac{1}{3} \\ \exp\left(\frac{3T-1}{2}\right)\varepsilon_{eq} & if & otherwise \end{cases}$$

(2.15)

(2.16)

$$D = \frac{1}{\varepsilon_{cr}}\int \dot{D}dt$$

Where p is the hydrostatic pressure, q is the Von Mises stress, ε_{eq} is the equivalent strain, ε_{cr} the critical strain, σ_1 the major principal stress and D is the damage. When the damage has reached unity in all through thickness integration points in the shell element, it deletes

the element. When unity is reached in one integration point, the stiffness will be set to zero to simulate a crack propagating through the element.

The elements that were used are one of the most used elements for collision analyses and the default element in LS-DYNA the four noded Belytschko-Tsay shell elements. It was used five through thickness integration points to properly use the RTCL damage [17].

2.2.2 Shipside scantlings

When sailing in the northern areas there is the possibility of sailing in to an ice cover. The ships are therefore often built according to some ice class rules. A usual and often used ice class rule are the Finnish-Swedish ice class rules that were made for the Baltic sea. The most used of the Finnish-Swedish ice class rules in the arctic areas such as the Northwest Passage are the IA. In this study there will therefore be studied the IA class and the effect of having a lower ice class the IC and have a ship without ice strengthening. The dimension used where gathered from actual tanker ships with a size at 150 000 tons.
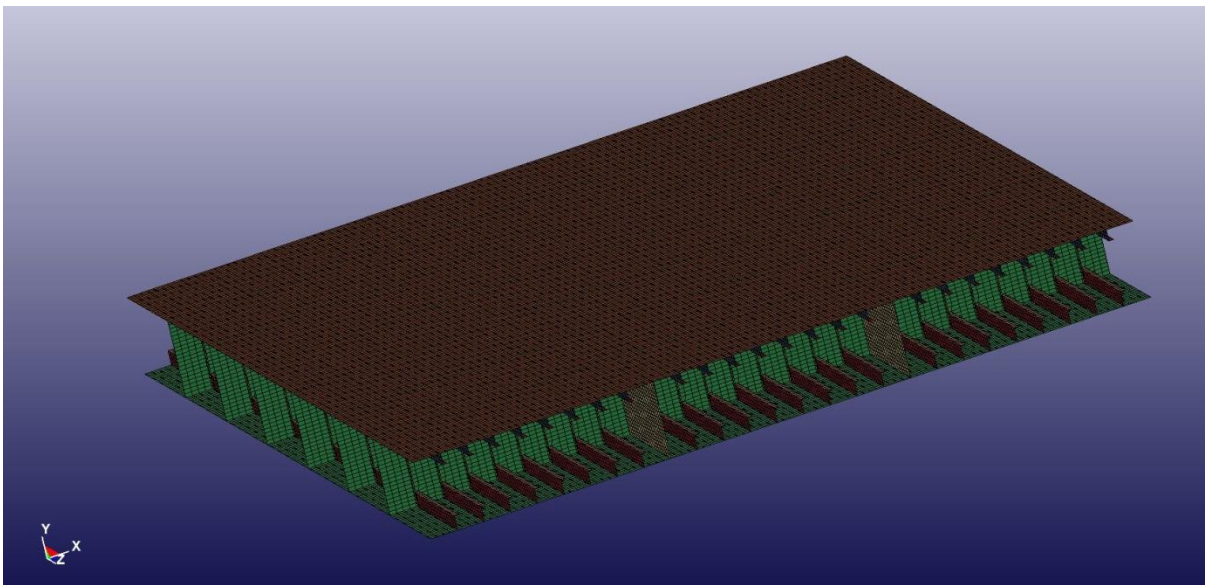
IA class

Web frame spacing	Web frame plating thickness	Stiffener spacing	Outer stiffener	Inner stiffener	Web frame stiffeners
3.36m	20mm	0.864m	L450x12/ 150x15mm	L350x12/ 150x15mm	FB200x12mm

Outer plate thickness	Inner plate thickness	Stringer spacing	Stringer plating thickness	Size of modeled field	Distance from inner to outer plating
30mm	18mm	7.8m	12mm	23.5x23.3m	2.41m

Table 6: IA Ice class hull dimension
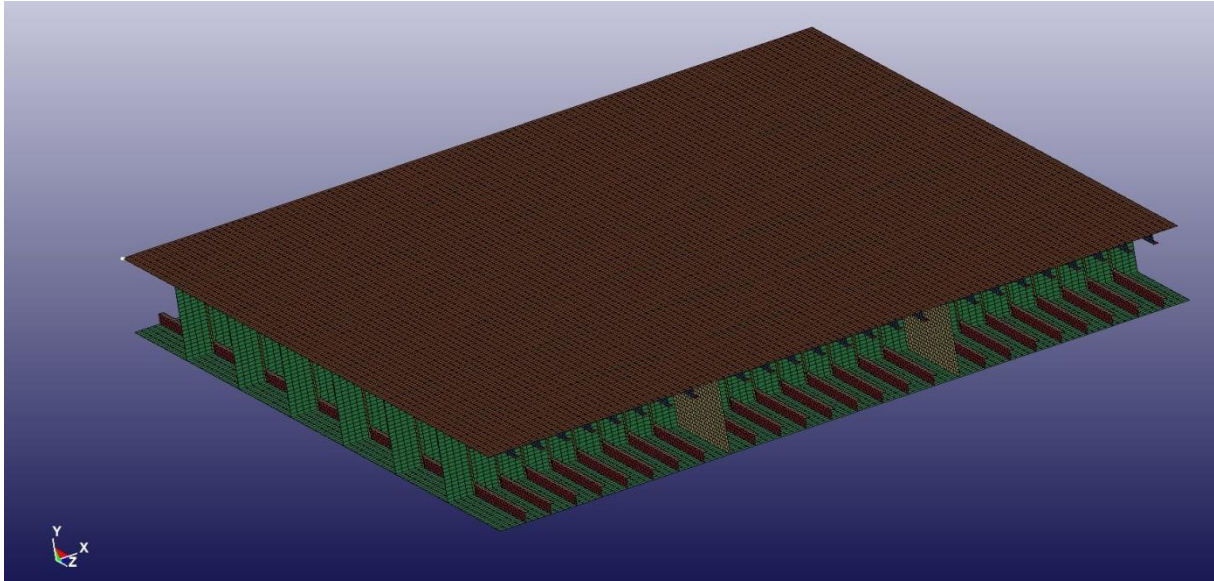
IC class

Web frame spacing	Web frame plating thickness	Stiffener spacing	Outer stiffener	Inner stiffener	Web frame stiffeners
3.36m	20mm	0.864m	L325x12/ 150x15mm	L350x12/ 150x15mm	FB200x12mm
Outer plate thickness	Inner plate thickness	Stringer spacing	Stringer plating thickness	Size of modeled field	Distance from inner to outer plating
21mm	18mm	7.8m	12mm	23.5x23.3m	2.41m

Table 7: IC Ice class hull dimension
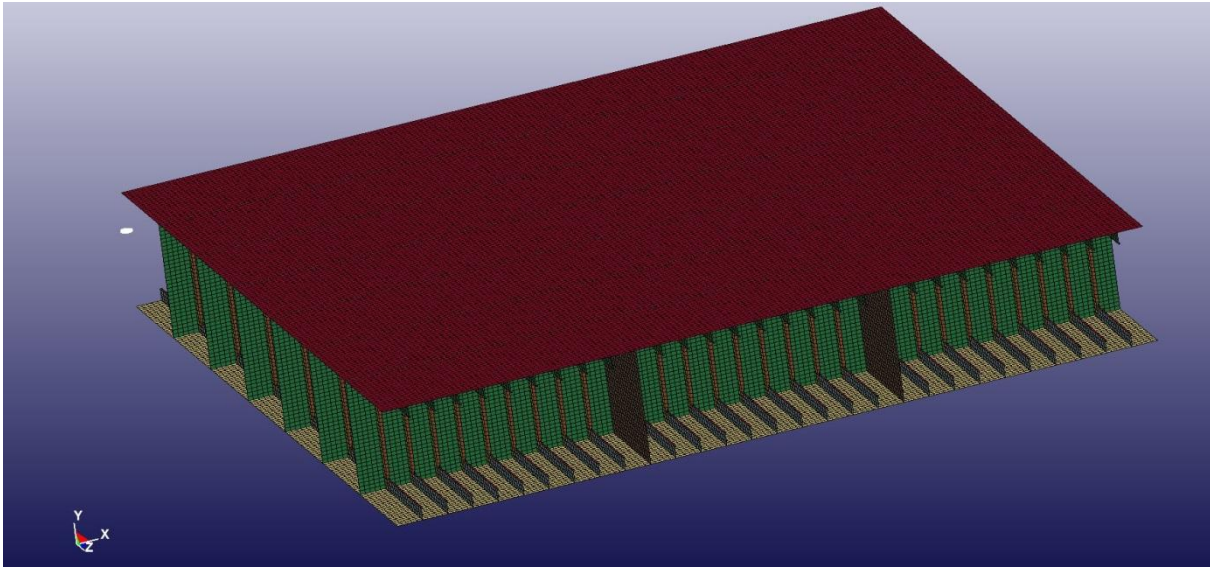
Non-ice class

Web frame spacing	Web frame plating thickness	Stiffener spacing	Outer stiffener	Inner stiffener	Web frame stiffeners
3.2m	15.5mm	0.8m	HP370x13mm (L370x13/ 53.5x38.4mm)	HP370x13mm (L370x13/ 53.5x38.4mm)	L250x11.5/ 100x15mm
Outer plate thickness	Inner plate thickness	Stringer spacing	Stringer plating thickness	Size of modeled field	Distance from inner to outer plating
17mm	15mm	8m	12mm	22.4x24m	3.6m

Table 8: The hull dimension of the ship without ice strengthening. The stiffeners are bulb stiffeners that has been simulated as equivalent L stiffeners for the use of shell elements

2.2.3 Mesh density

For the ship the element size was kept below 10 times the thickness for any part of the shipside. To avoid getting some elements that were much smaller than others and thereby demand a smaller time step there were avoided to have transitions in the mesh from one part to another, such as from the stiffener flange to the stiffener web and stiffener web to the plating. The effect of this was that the outer plating has an element size of about 6 times the thickness of the plating.
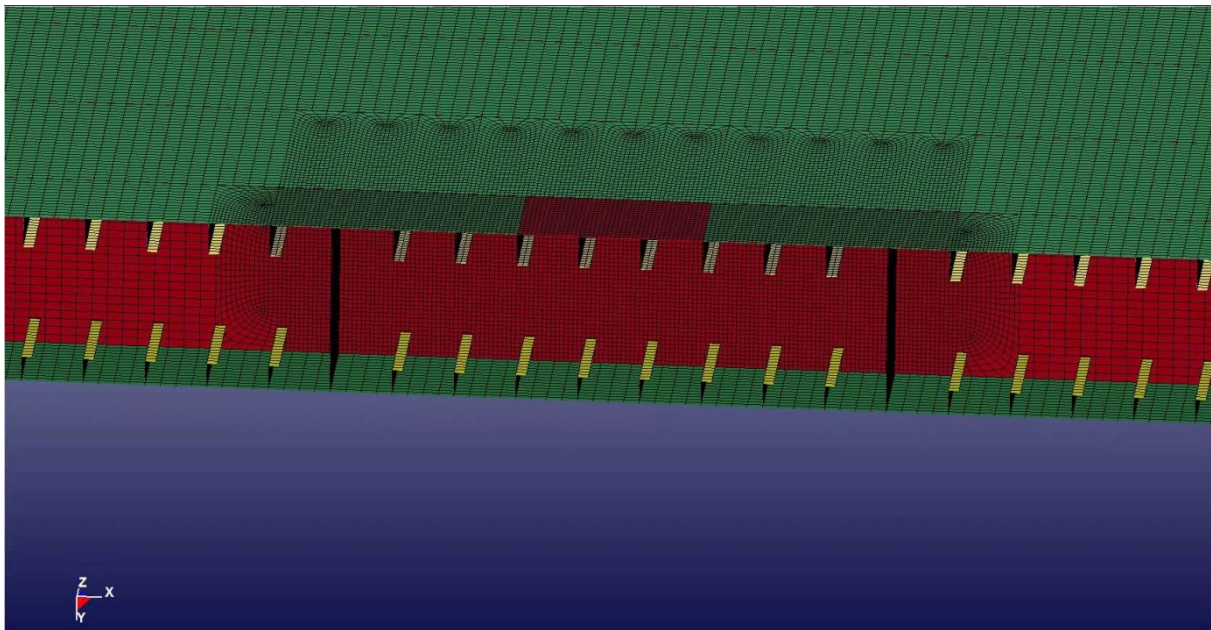
Figure 12: Cut through 1A shipside in the collision area, showing the refined mesh density

There was done an analysis with different mesh density in the contact area to find the effect of the mesh density. There every part that was given large deformation due to the collision was tested with various mesh density. This caused the web frame between two stringers, the stringers between two web frames, the stiffeners on the web frame in this area, stiffeners and plating between the web frames and stringers to be tested with changed mesh density. In this analysis there were used element sizes from 2 to 10 times the thickness of the outer plating.

The result from this analysis showed that the element must be at most 7 times the thickness of the element.

To save computing time the analysis were run without refining of the mesh at about 6 times the element thickness, so no transitions where needed.

2.3 Model setup

The whole modeling, setup of the analysis, executing of the analysis and gathering of result files was done by the matlab script shown in appendix A. With the matlab script any of the input values can be varied and run in an "for" loop to analyze the effect of these variables.

The models where modeled separately in Patran where the boundary conditions and meshing also were done. This was done with a Patran control file that were created and run in Patran by the matlab script in appendix A. This way there is easy to change any of the dimensions. The meshed models were exported to LS-Prepost and put together so that the iceberg radius was nearly touching the middle of the outer plating of the shipside. This was done by a LS-Prepost control file that were lunched by matlab, seen in appendix C. The layer of elements furthest away from the shipside in the iceberg was given a rigid material, seen in blue in Figure 13. This layer was kept from moving in any direction but normal to the shipside. The nodes in this layer were given a moving boundary condition, so that the iceberg was pushed towards the ship at a constant speed of 10 m/s seen in Figure 13.
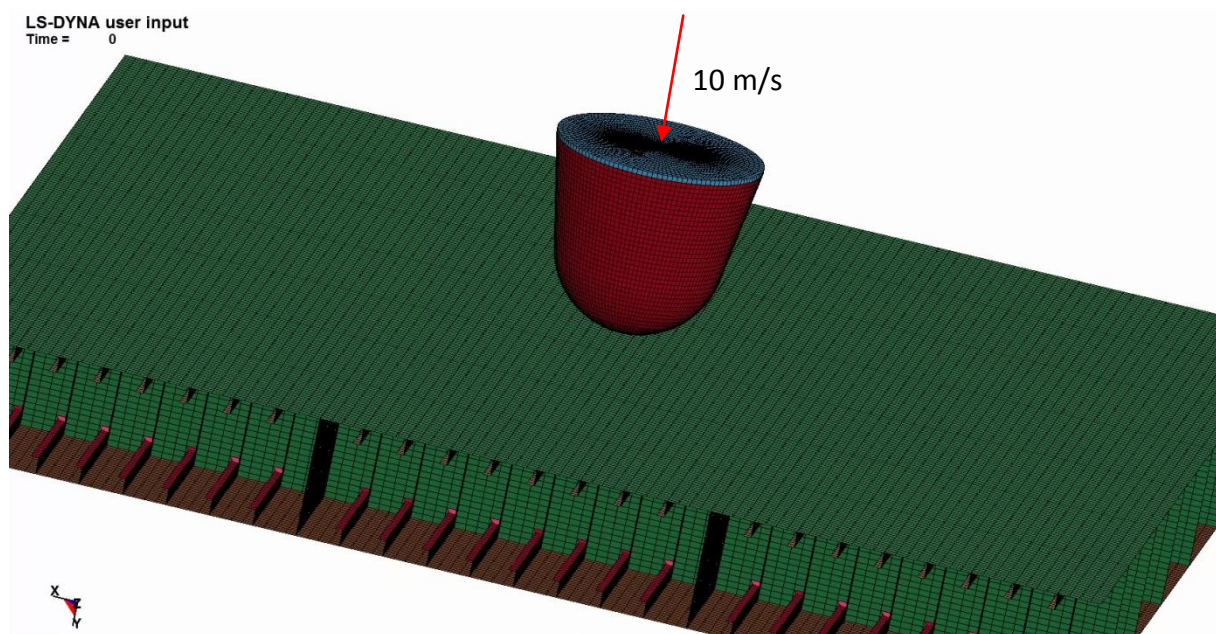
Figure 13: Model

All edges of the section of the shipside that would have been connected to the rest of the ship where clamped as marked by the white dots in Figure 14.
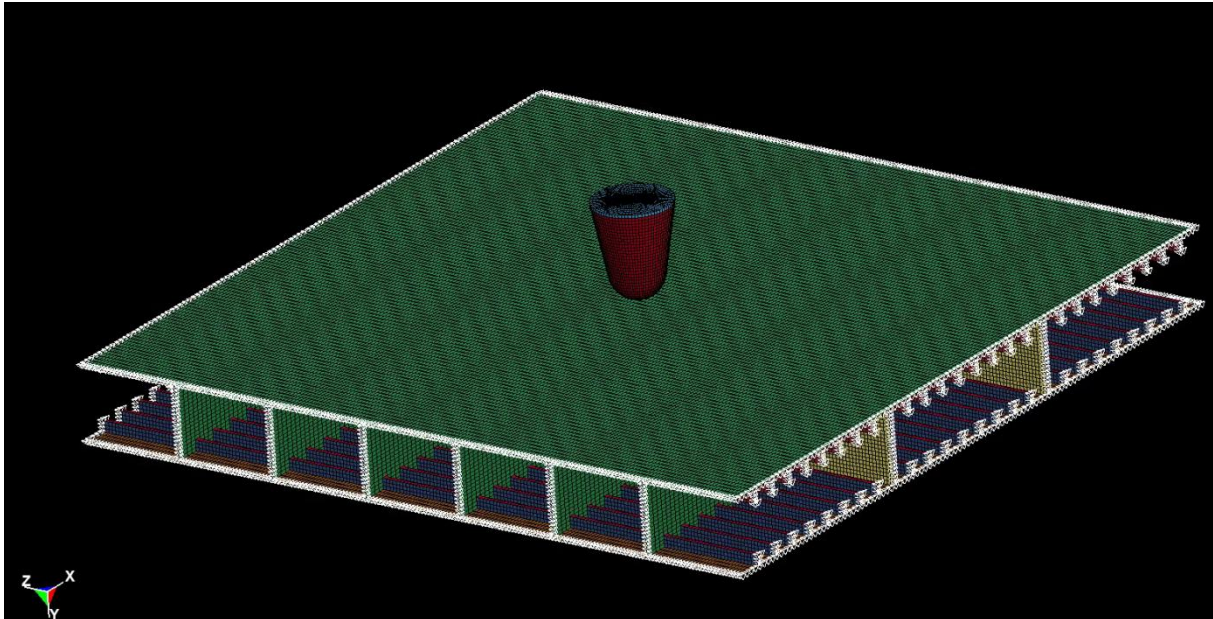
Figure 14: Clamped boundary conditions

The forces in the boundary did get some dynamic oscillations, but were assumed not to be influencing the result so much since the oscillations cannot be seen in the contact force. As seen in Figure 15.
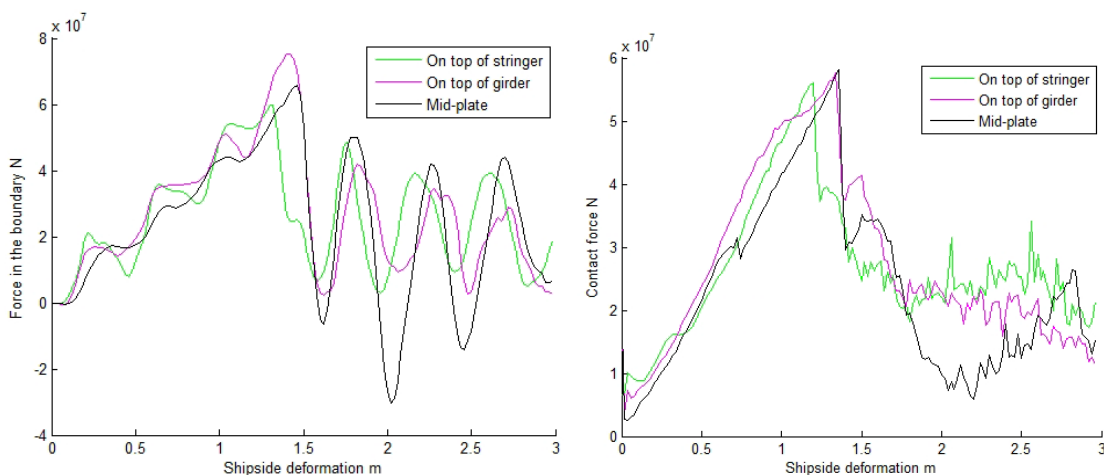
Figure 15: Checking boundary conditions by comparing contact force with the force in the boundary

The result was gathered from the ncforc output file where all coordinates and nodal reaction forces and pressures were read. These data was then analyzed by another matlab script seen in appendix B. The values gathered from the ncforc file were used to calculate the values for all graphs seen in chapter 3. The explanation of the calculations is found before each graph in chapter 3.1.1.

3 Results and discussion

3.1 Integrated simulation

3.1.1 Without ice strengthening

The impact force was measured by summarizing the reaction force in each node on the iceberg. Then the deformation in the shipside was found by reading all the coordinates of the iceberg and finding the coordinate of the node nearest the shipside. This will give the maximum deformation of the shipside. The deformation of the iceberg was found by using the time step and the speed to find the distance traveled, and then subtract the deformation of the shipside.
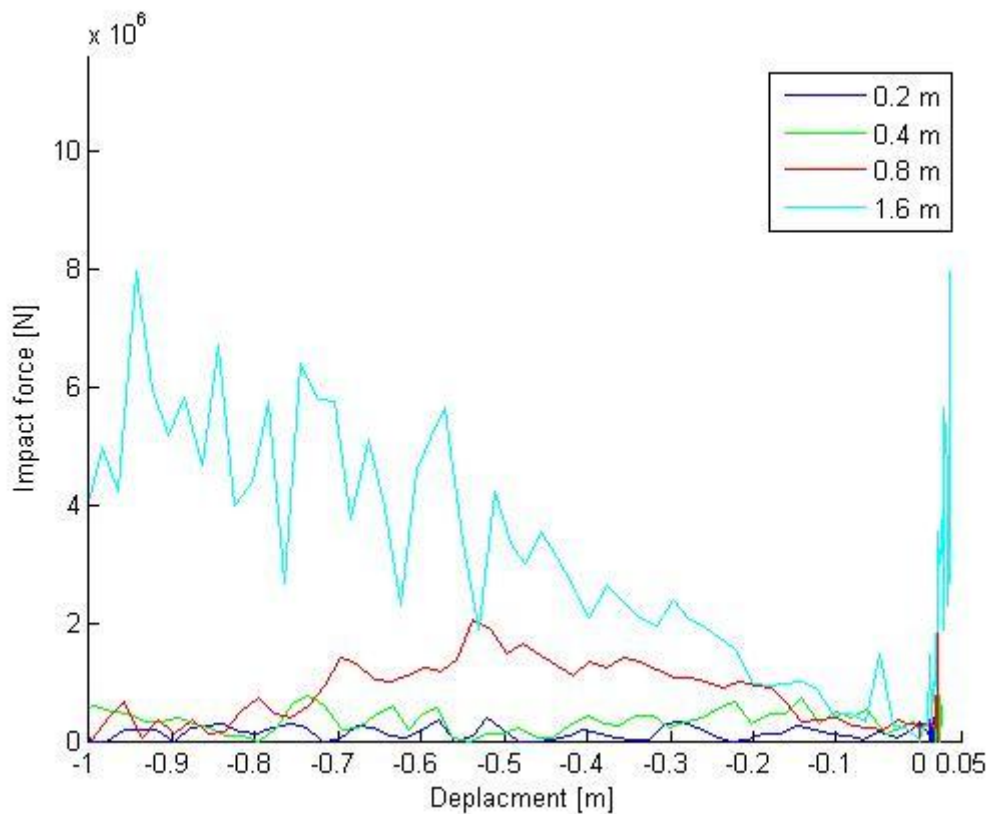
Figure 16: Impact force against displacement for the ship without ice strengthening. Ship to the right and iceberg to the left

Figure 16 shows the impact force from four different icebergs with the ship without ice strengthening. The impact force for all four icebergs were increasing to a point that was

about an $\frac{1}{4}$ of the radius for the smallest iceberg to about $\frac{3}{5}$ for the largest iceberg and then decreasing to almost zero again. The increasing in impact force was as expected because of the increase in the contact area of the iceberg will give its contribution.

In the analysis done by Zhenhui Liu [10] with the same ice material model there was also found a drop in the impact force initially, that relates to the contact with the stringers. In this thesis the drop happens with different diameter of the contact zone for the different iceberg shapes. And there were not possible to see a direct connection with the stiffener, stringer or web frame spacing.

The analysis done by Storheim and Kim [12] which also uses the same material model for ice there were also shown a drop in force initially. Here it can be related to the element size and the force was dropping because of the eroding of a layer of elements. This thesis also shows drops in impact force due to the eroding of many elements in the same time step, but this effect only gave small waves in the impact load and does not influence the peak pressure much.

The angle of the side of the iceberg will change during the impact until the whole radius was crushed, and then the angle will be constant during the deformation of the cone behind. This gives that the confinement due to the pressure from the ice behind the contact area will decrease until the deformation reaches the cone. The confinement may be the reason for the impact force to decrease.

As the iceberg was crushed the angle between the iceberg side and the shipside increases and thereby decreasing the confinement of the edges of the iceberg. The difference is shown in Figure 17 and Figure 18. When the elements on the edges were eroded it created new edges that will have low confinement and then be eroded. The movement of the point where the impact force decreases could be caused by the deflection of the shipside. The shipside deflects more with the larger radius icebergs and thereby decreasing the angle between the shipside and the side of the iceberg.
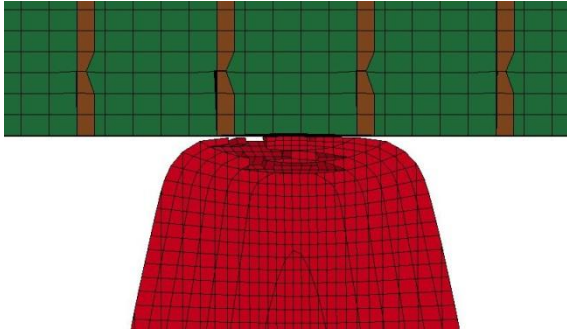
Figure 17: High confinement in the iceberg edges
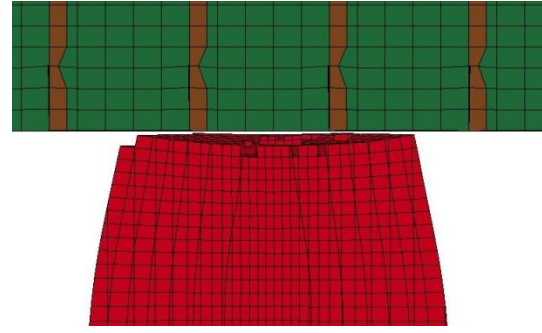
Figure 18: Low confinement in the iceberg edges due to increased angle with the shipside

In between the states in Figure 17 and Figure 18 there is failure of multiple elements in an angle from the side to the middle of the iceberg, shown in Figure 19.
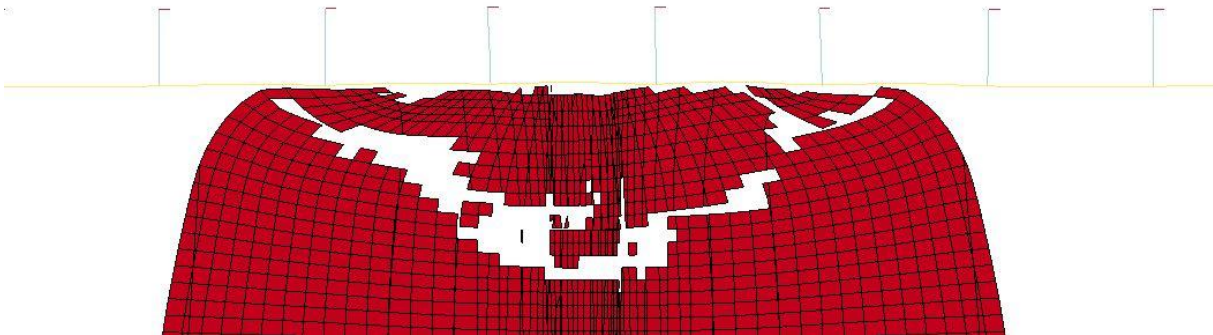
Figure 19: Fracture at peak impact force for 1.6 m radius iceberg on the ship without ice strengthening

This was likely caused by the large strains at the middle has caused a shear fracture when trying to spread the force outwards.
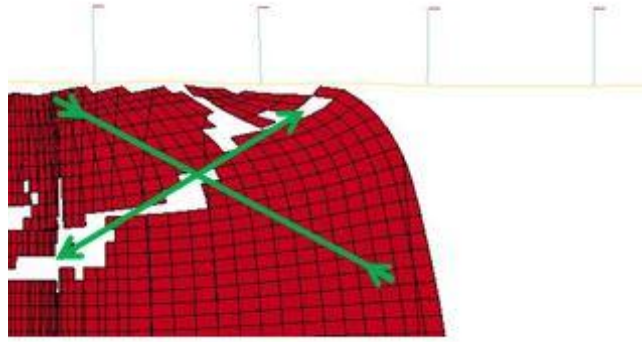
Figure 20: Forces in the shear fracture of the iceberg

The elements that where eroded was in a condition of compression in one direction and in tension in the other, as shown in Figure 20. This will cause the second invariant of deviatoric stress tensor to become larger in the Tsai-Wu yield surface equation (2.1), and then the equivalent strain equation (2.8) also gives higher values because the compression and tension in the elements. This causes the elements to fail.

The absorbed energy was found as the area under the displacement impact force graph in Figure 16.
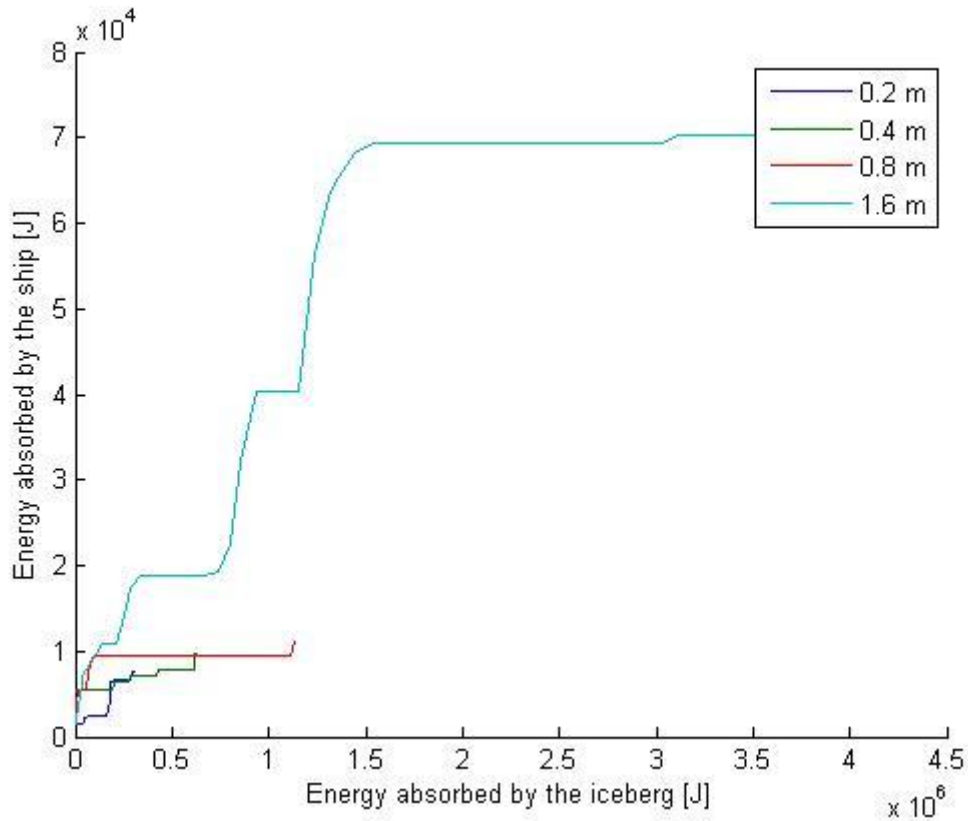
Figure 21: Distribution of energy between ship and iceberg for the ship without ice strengthening. Note the difference in the scale between the x-axis and the y-axis.

The energy distribution curves for the two largest radius icebergs has about the same gradient and the two smaller has a much lower gradient. The iceberg was deforming the ship at first but at a point the ship becomes stronger than the ice and stops absorbing energy. As seen in Figure 16 there were no more deflection of the shipside after the point of highest force and thereby the shipside was not absorbing more energy.

The area was found by taking the resultant force in each node and then divide it by the pressure in the same node, then summarized to find the total nominal area. The average pressure was found by dividing the impact force by the area. There was plotted a result point from each time step.
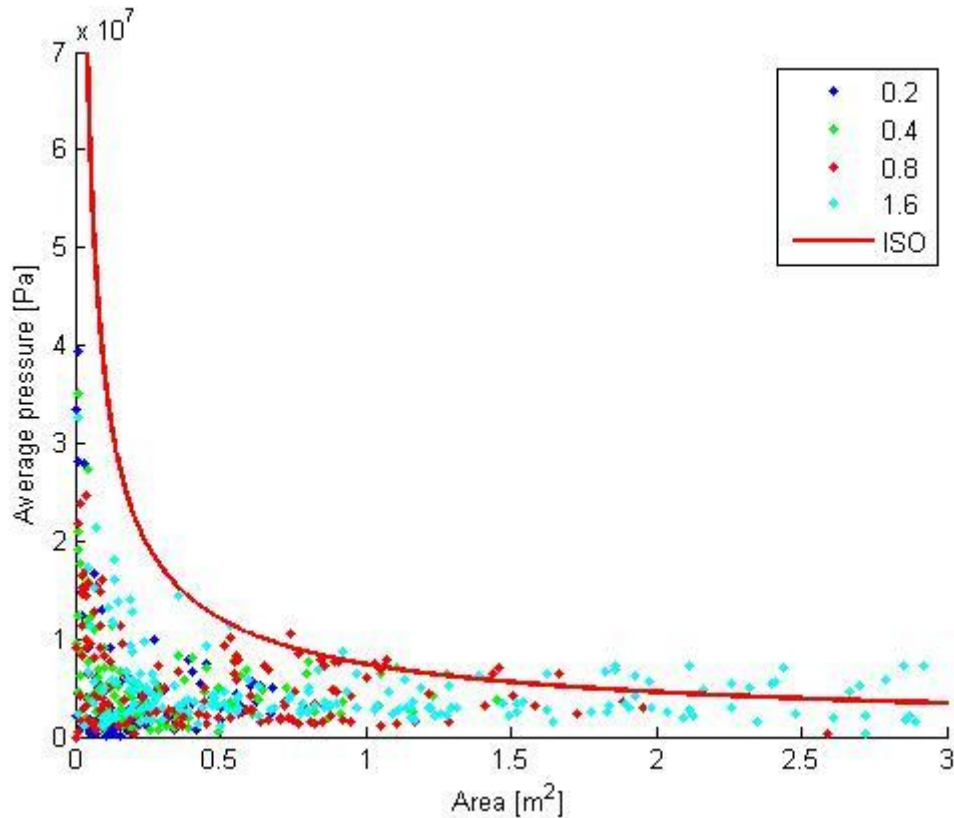
Figure 22: Average pressure-nominal area relationship for the ship without ice strengthening

The average pressure over nominal area was increasing for larger radius and should be seen in context with that the confinement of the ice material at a given contact area will be greater for a larger radius. The curve from the ISO-standard [5] seems to coincide with the upper band of the data from the largest icebergs in the analysis in the areas smaller than 0.7 m^2. For areas larger than 0.7 m^2, the curve from the ISO-standard seems to under estimate the upper band of the pressures. The curve from the ISO-standard is based on many ice experiments and therefore likely to be more accurate than the ice material model. The higher pressures were likely caused by the waves in the impact force due to the new collision after the eroding of a layer of elements.

The total energy seen in Figure 23 was found by adding the energy absorbed by the ship with the energy absorbed by the ice at each time step. The values for the energy absorbed by the ship and the ice was found like for Figure 21.
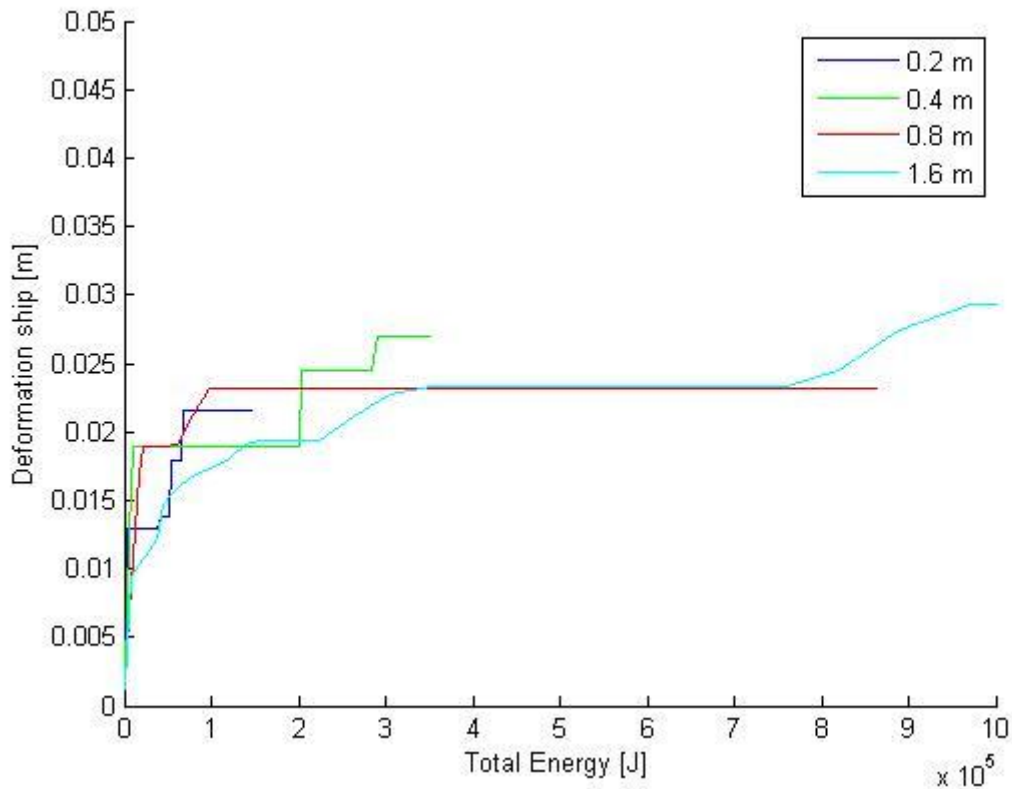
Figure 23: Total collision energy - deformation of shipside for the ship without ice strengthening

No large differences can be seen in Figure 23. At 0.1 MJ of energy the 0.2 m radius iceberg and the 0.8 m radius iceberg gave the most deformation of the shipside followed by the 0.4 m and then the 1.6 m in radius iceberg. The reason for that the smallest iceberg give larger deformation at lower energy were that they have to move less of the shipside to cause deformation. All icebergs but the largest one gave about the same deformation at the end.

36

3.1.2 1C ship
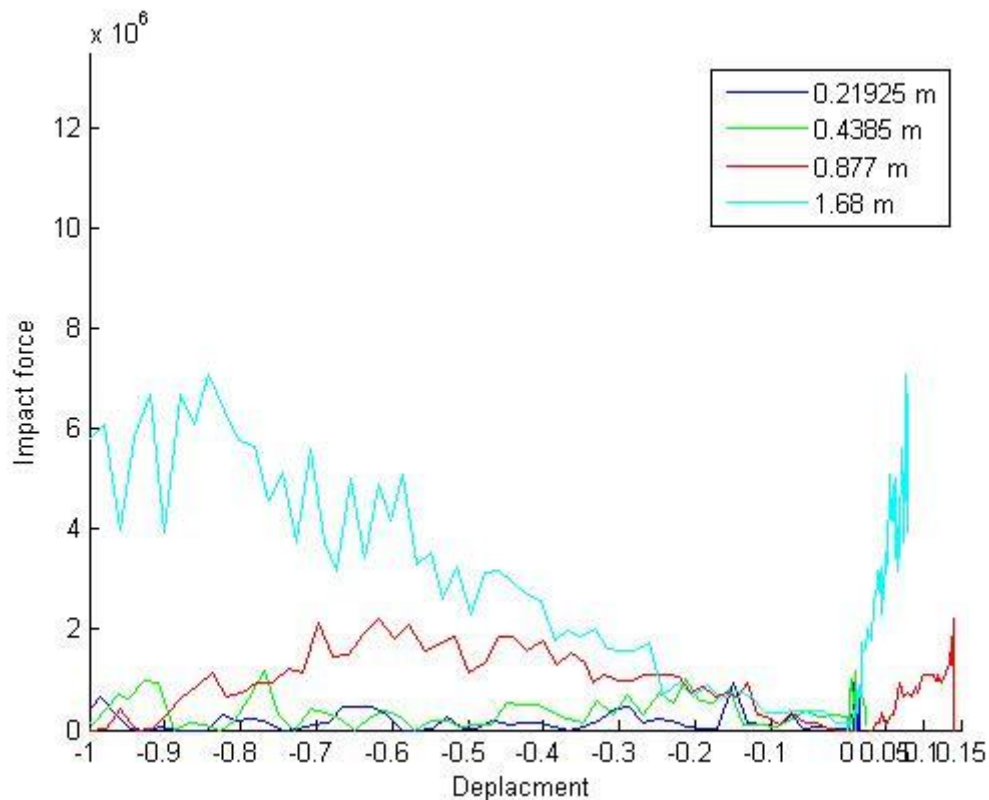
Figure 24: Impact force against displacement for the 1C ice strengthened ship. Ship to the right and iceberg to the left

The same tendency as for the ship without ice strengthening the forces is increasing to a point and then decreasing again. The point is moving with increased radius, but the maximum value is closer to half the radius. The deflection of the shipside has increased by about a factor of three from the ship without ice strengthening. This may be caused by the increase in stiffeners spacing and web frame spacing.
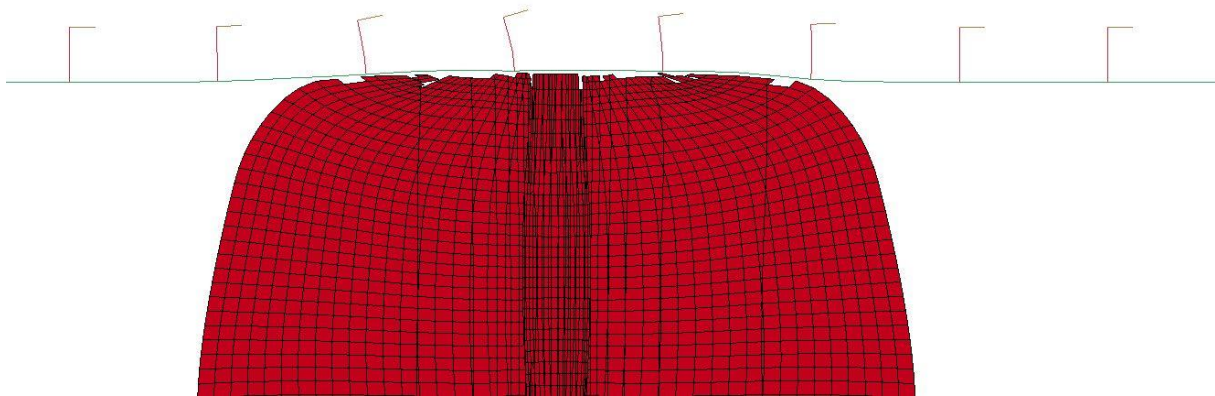
Figure 25: Deformation at maximum impact force with 1.68m in radius iceberg on the 1C ship
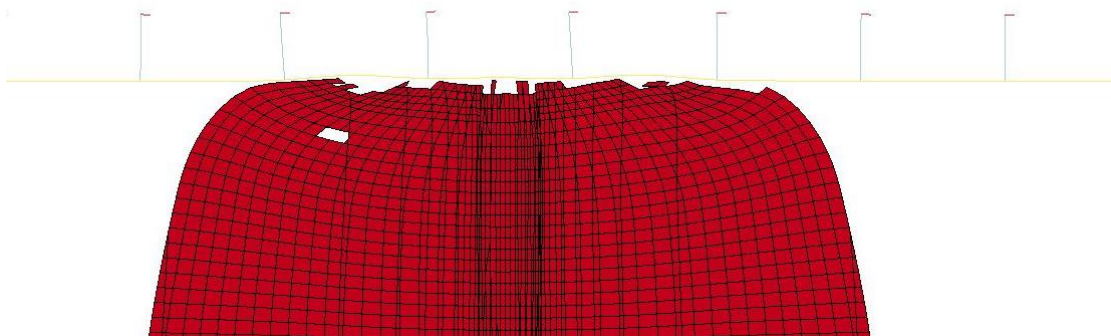
Figure 26: Deformation at maximum impact force with 1.6 m radius iceberg on the ship without ice strengthening

The deformation is shown for the largest iceberg for the ship without ice strengthening and for the 1C ice strengthened ship. The deflection in Figure 26 is barely visible and in Figure 25 for the 1C ship there is a clear bending of the stiffeners.
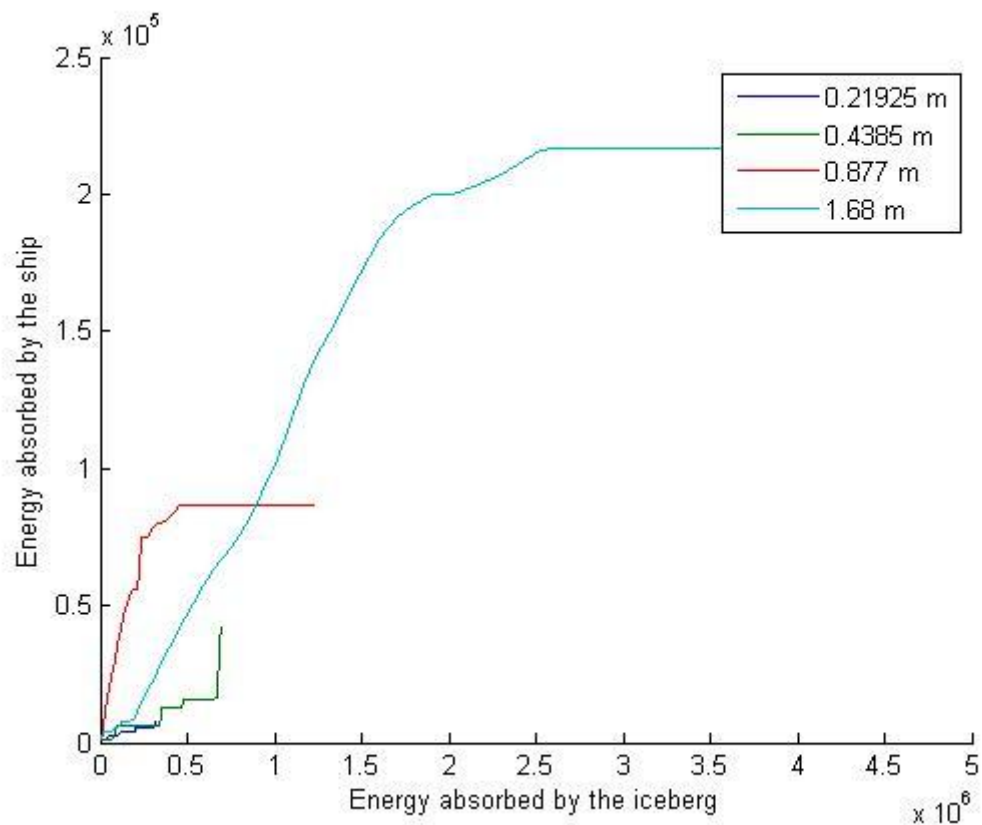
Figure 27: Distribution of energy between ship and iceberg for the 1C strengthened ship. Note the difference in the scale between the x-axis and the y-axis.

As seen in Figure 27 the distribution of energy between the two smallest icebergs and the shipside has the same gradient. Initially the largest iceberg also had the same gradient but it increases and becomes much larger than for the two smallest icebergs. The second largest iceberg which has a radius corresponding to a fourth of the web frame spacing has a far higher gradient than the others and does nearly as much damage to the ship as it does on itself.
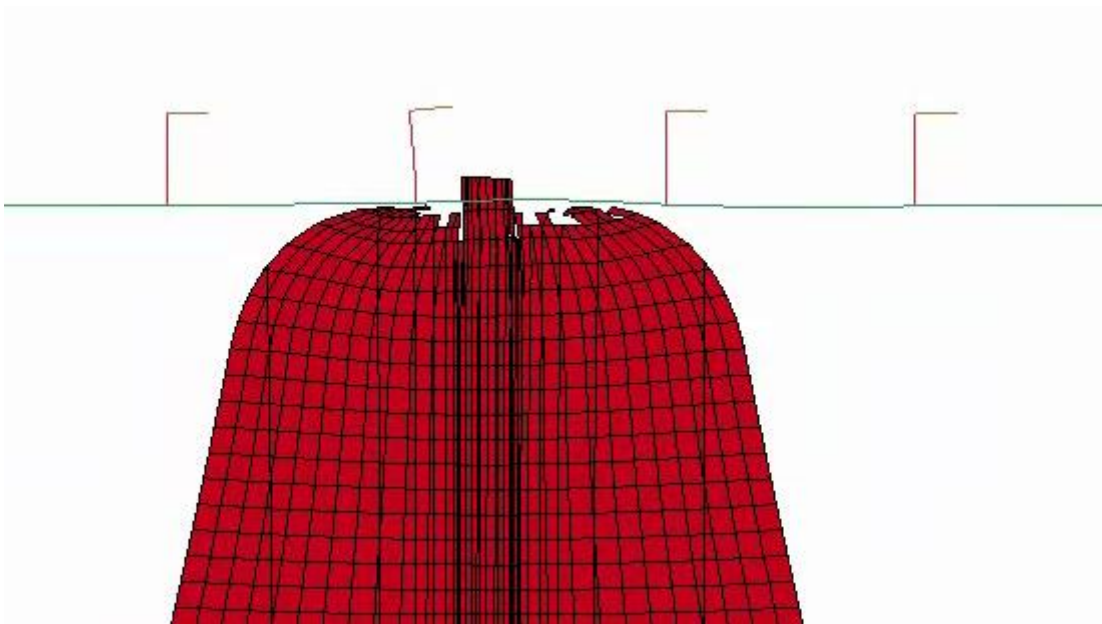
Figure 28: Simulation error with the iceberg with 0.877 m radius

This high gradient was caused by an error in the simulation. The deformation in the analysis was read of wrong because of the elements in the middle was going through the elements in the ships plating without rupture of the shipside, as seen in Figure 28.

The energy absorbed in the shipside were larger for this ship than for the ship without ice strengthening because of the larger deformation as shown in Figure 25 and Figure 26 .
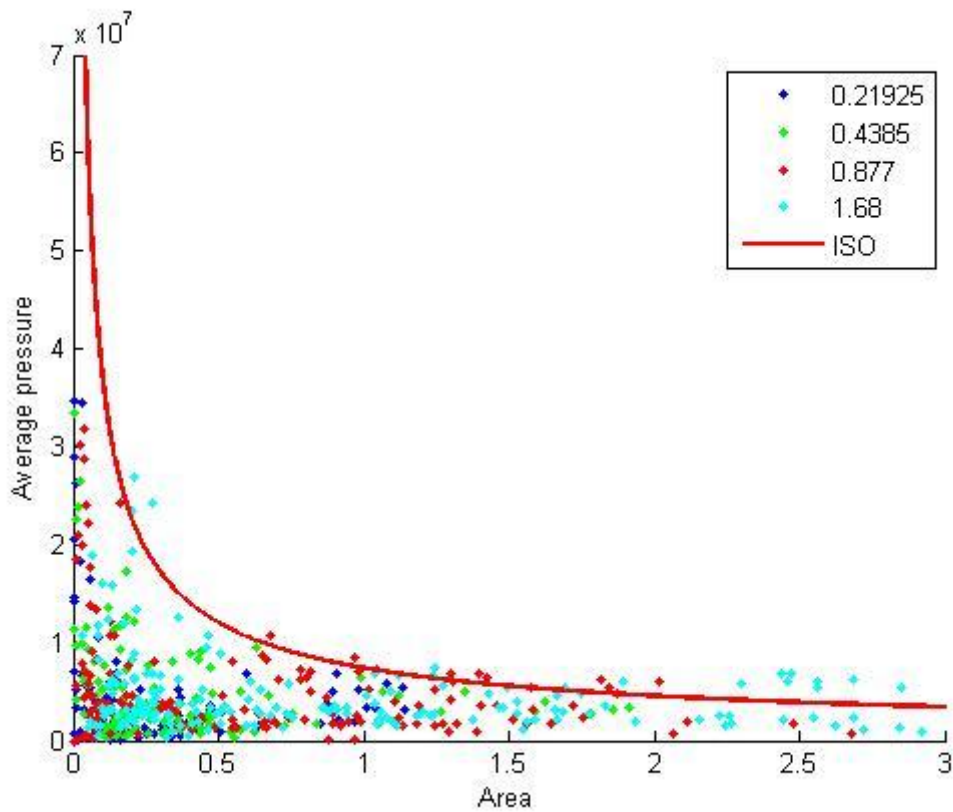
Figure 29: Average pressure-nominal area relationship for the 1C ice strengthened ship

For the 1C reinforced ship there is the same tendency as for the non-reinforced ship. For higher areas the pressure goes above the curve from the ISO-standard [5]. The pressures at smaller areas was also a bit higher than in the non-reinforced ship, which was likely caused by the thicker plating have larger bending resistance.
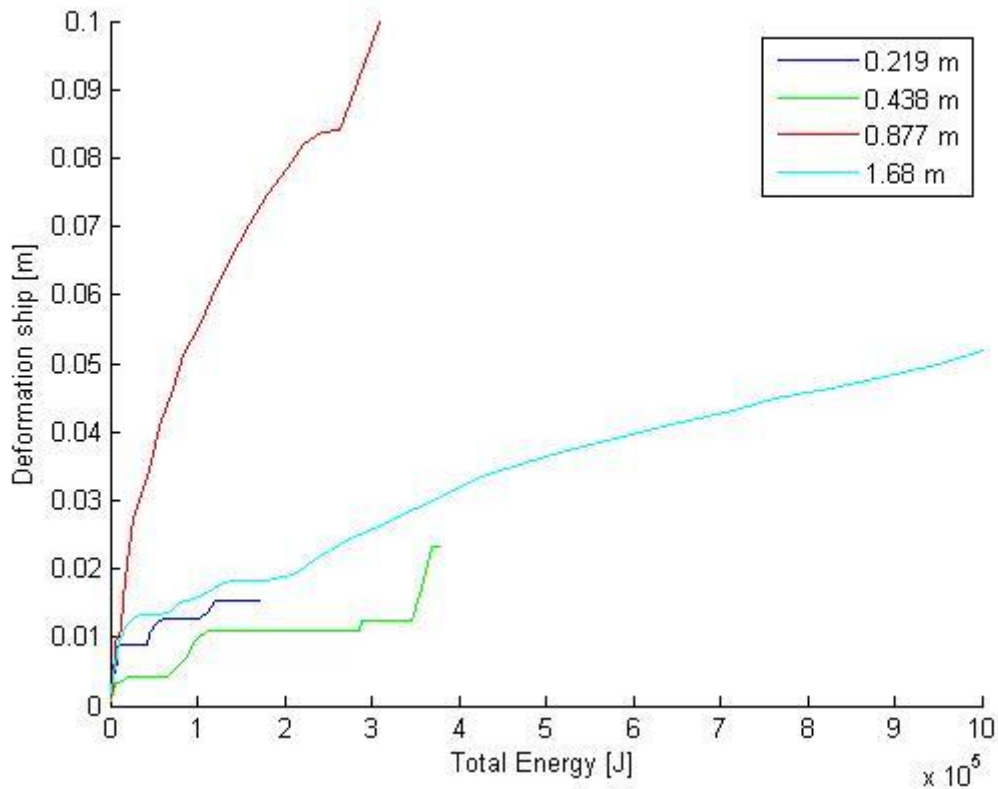
Figure 30: Total collision energy - deformation of shipside for the ship with ice class 1C

Figure 30 compared with Figure 23 shows that the shipside needs more collision energy to deform than for the ship without ice strengthening. The 0.877 m in radius iceberg gave much larger deformation than the rest of the icebergs as also seen in the other graphs, but was not correct as discussed below Figure 28. The 1.68 m iceberg gave about the same deformation for any collision energy as against the ship without ice strengthening in Figure 23. The smaller iceberg gave less deformation than against the ship without ice strengthening, likely because of the thicker plating was harder to bend.

3.1.3 1A ship
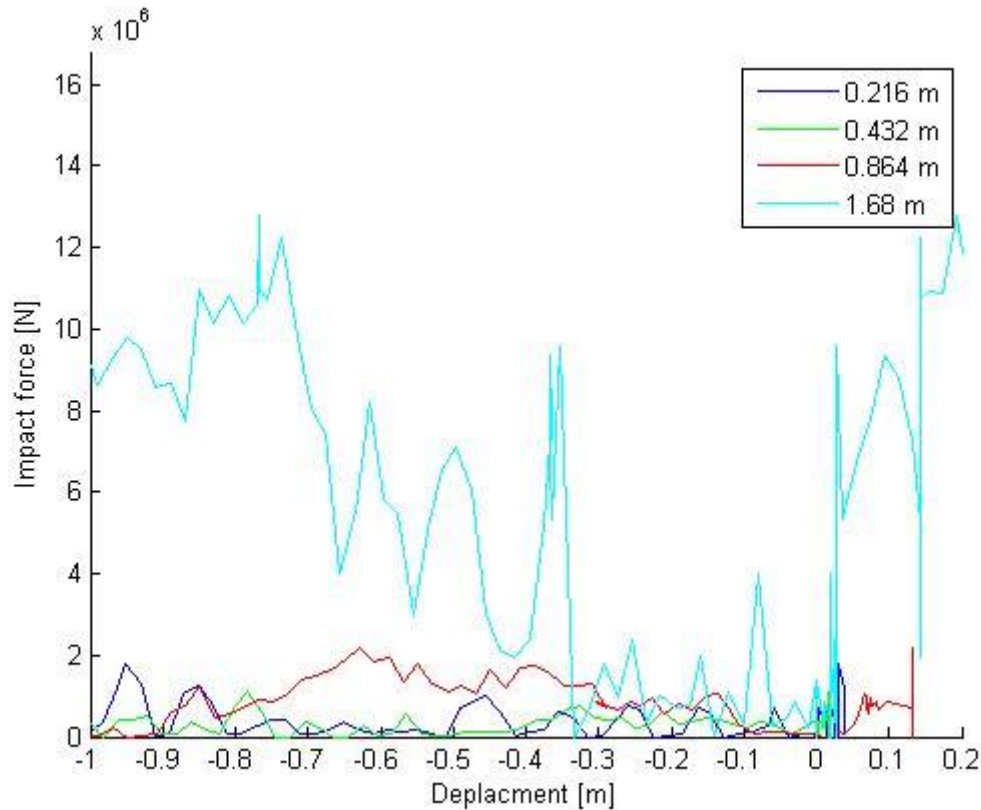
Figure 31: Impact force against displacement for the 1A ice strengthened ship. Ship to the right and iceberg to the left

The maximum value for the impact force was moving closer to the size of the radius of the icebergs than on the other ships. The impact force for the two smallest icebergs has a wave like appearance with wavelength equal to the element size due to erosion. The second largest icebergs give much higher deflection of the shipside than the other icebergs, but this was again caused by the elements in the middle was going through the ships plating as shown in Figure 28. The largest iceberg gave much larger deformation of the shipside and has very large waves in the force that do not correspond to the element size.
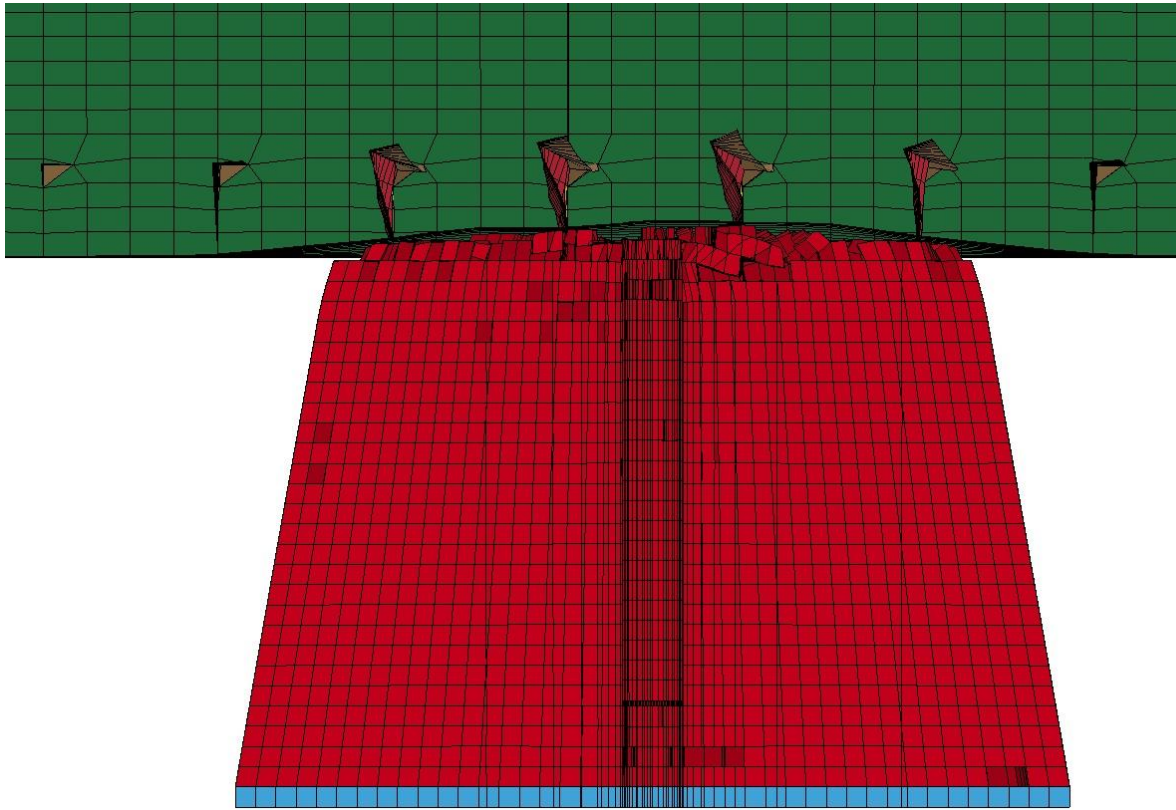
Figure 32: Deformation of 1.68 m radius iceberg on the 1A ice strenghtened ship from the second zero impact force point

As seen in Figure 32 the element behind the ones in front are eroded away and thereby the ones in front are unsupported and cannot take any load. This happens over and over again on this iceberg and causes a new collision for each time. This causes plastic deformation on the shipside for each collision. This behavior was not unlike what could be expected by ice when crushing.
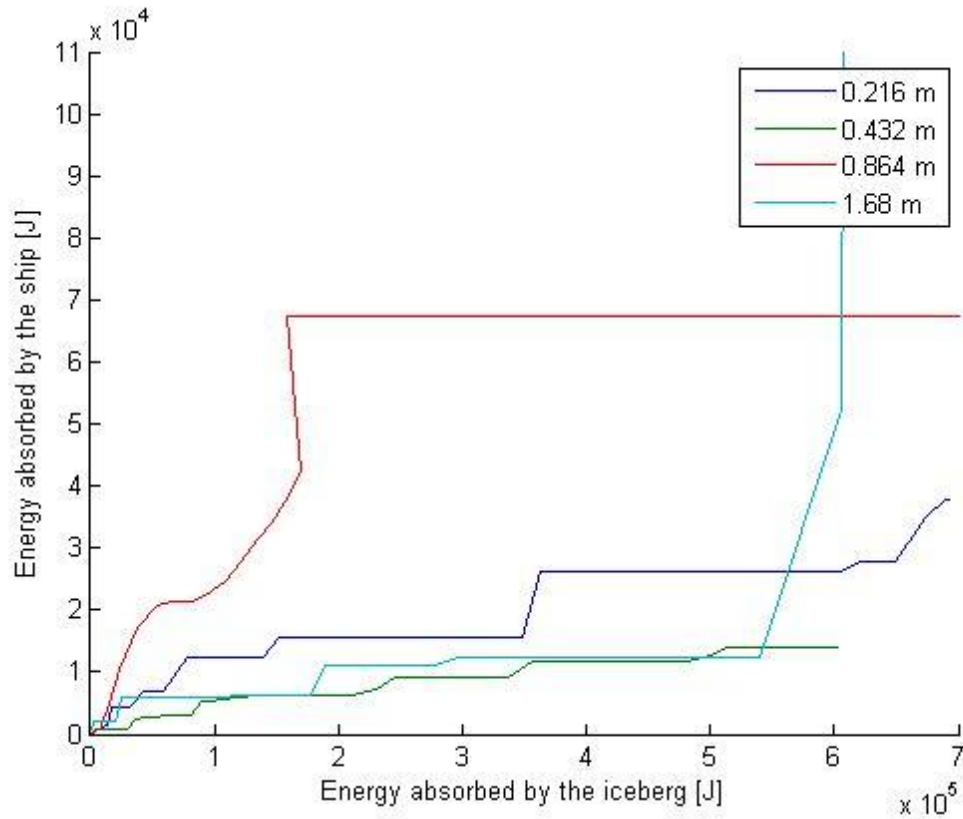
Figure 33: Distribution of energy between ship and iceberg for the 1A strengthened ship. Note the difference in the scale between the x-axis and the y-axis.

The energy absorbed by the smallest iceberg was higher than for the second smallest. This was likely because of that the elements were too large for this iceberg causing large waves in the collision force for each eroding layer of elements seen in Figure 34.
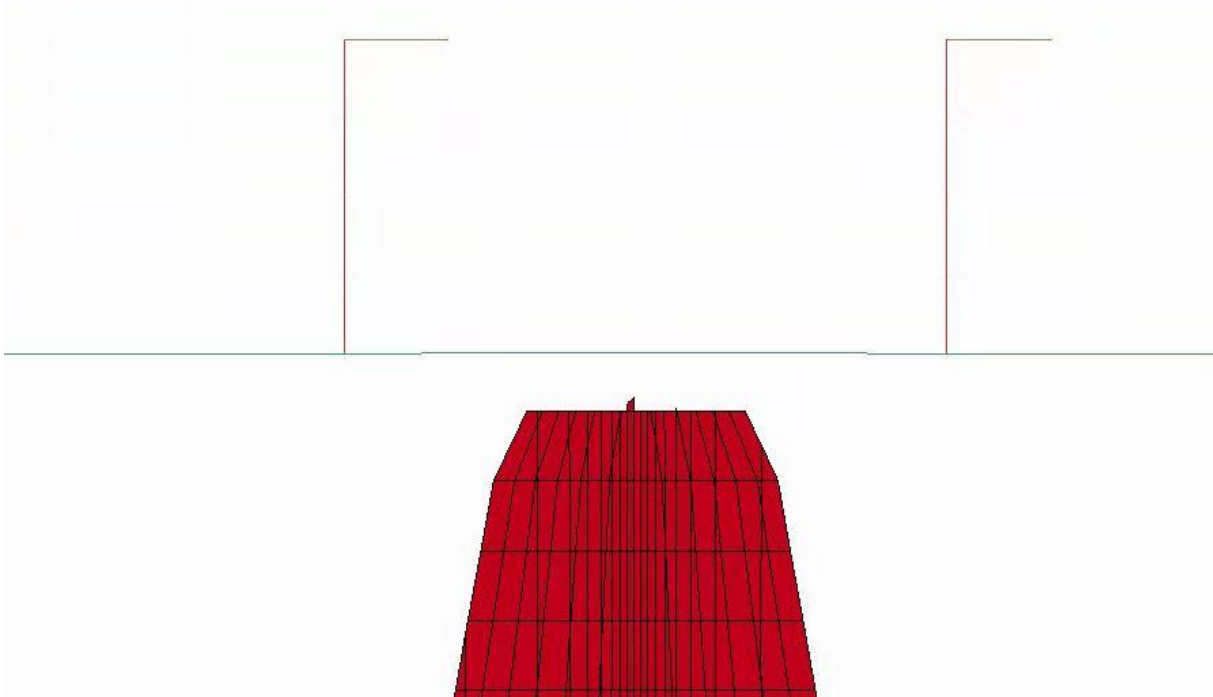
Figure 34: Eroding of elements in the 0.216 m in radius iceberg on the 1A ice strengthened ship

The smallest iceberg in the collision with the 1A ice strengthened ship was eroding a whole layer of elements when colliding and creating high impact force at contact followed by zero impact force when the elements was deleted. This iceberg should either be simulated with smaller elements or less speed to avoid the deleting of elements at first time step after contact.
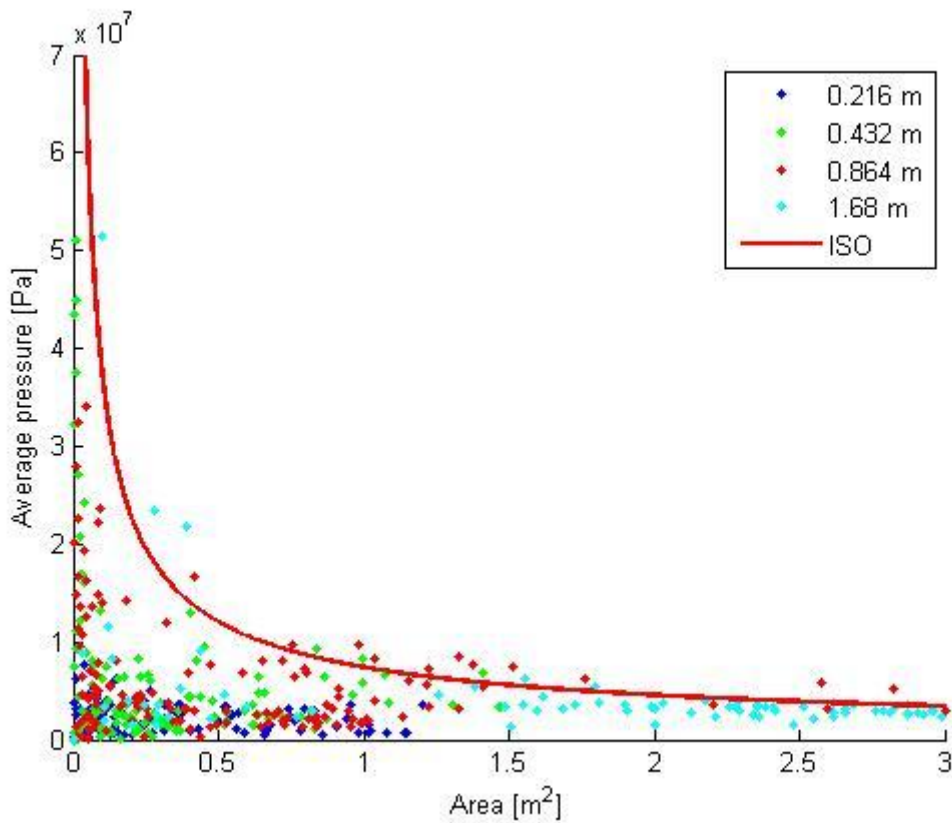
Figure 35: Average pressure-nominal area relationship for the 1A ice strengthened ship

The pressure-area relationship has been altered for the smallest iceberg. It takes much lower pressures against the 1A ship than for the weaker ships, except for the pressure at near zero area was higher. This was because the plating of the ship was so thick that they were nearly not deflecting and when the first element hit the plate creating high pressures because of the small area. The other iceberg experiences as much pressures against the 1A ship as against the other ships.
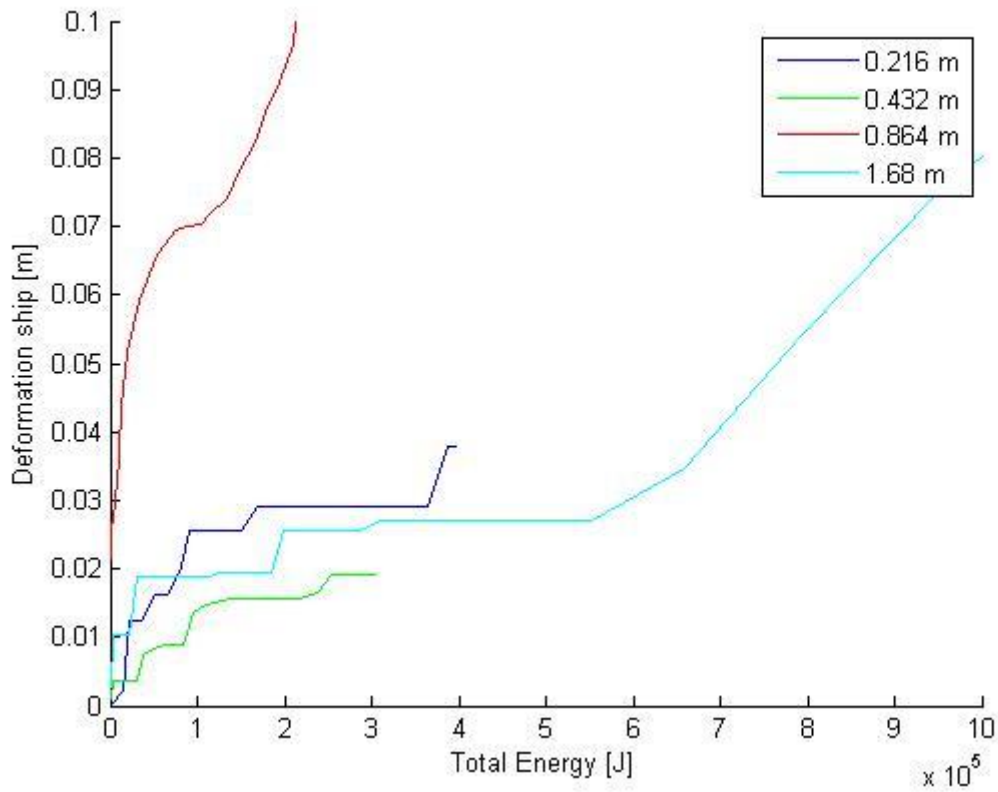
Figure 36: Total collision energy - deformation of shipside for the ship with ice class 1A

Seen in Figure 36 the smallest iceberg gave a higher deformation and higher total collision energy than the second smallest iceberg. This was as explained bellow Figure 34 caused by repeated collisions after eroding of a layer of elements.

3.1.4 Angle check for the ship without ice strengthening
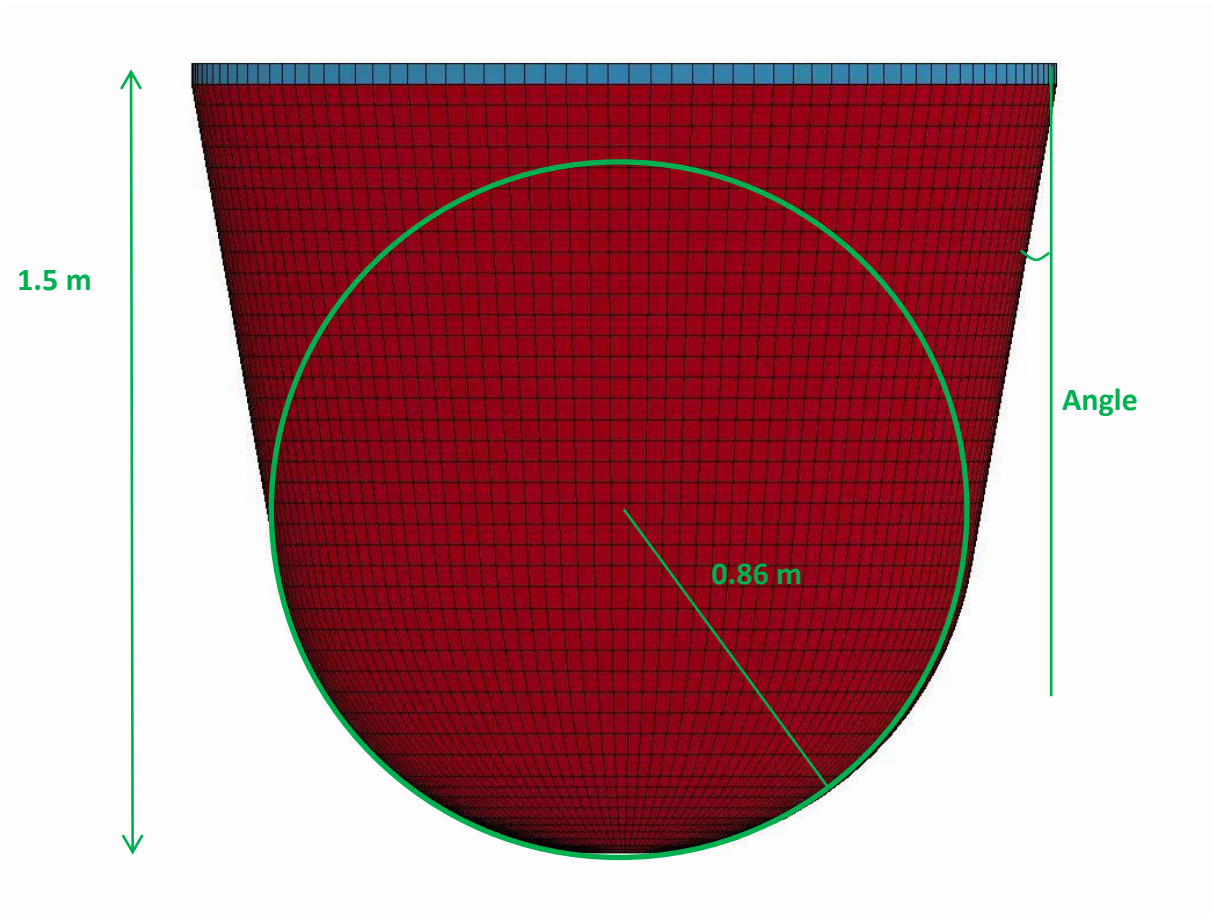
Figure 37: Angle description

This analysis was done to look for the effect of confinement by varying the angle that was described in Figure 37.
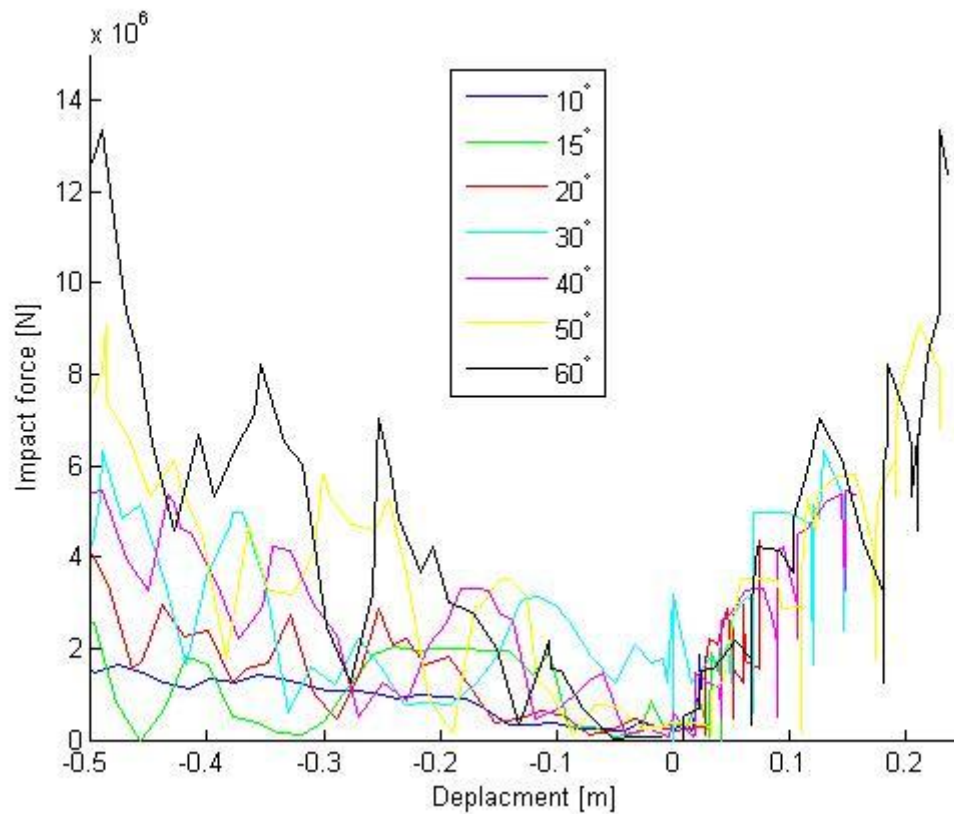
Figure 38: Impact force against displacement for different cone angles on the ship without ice strengthening. Ship to the right and iceberg to the left.

When changing the angle of the cone behind the radius the impact force seems to be rising with higher angle, but the area are also larger for any value above 0.1 m deformation. The 30° iceberg has the highest impact force before 0.1 m deformation followed by the 40° and then the 50° iceberg. The impact forces for the two largest icebergs has the same large waves as the impact force for the largest iceberg in Figure 31 caused by repeated fracture of the iceberg due to shear force.
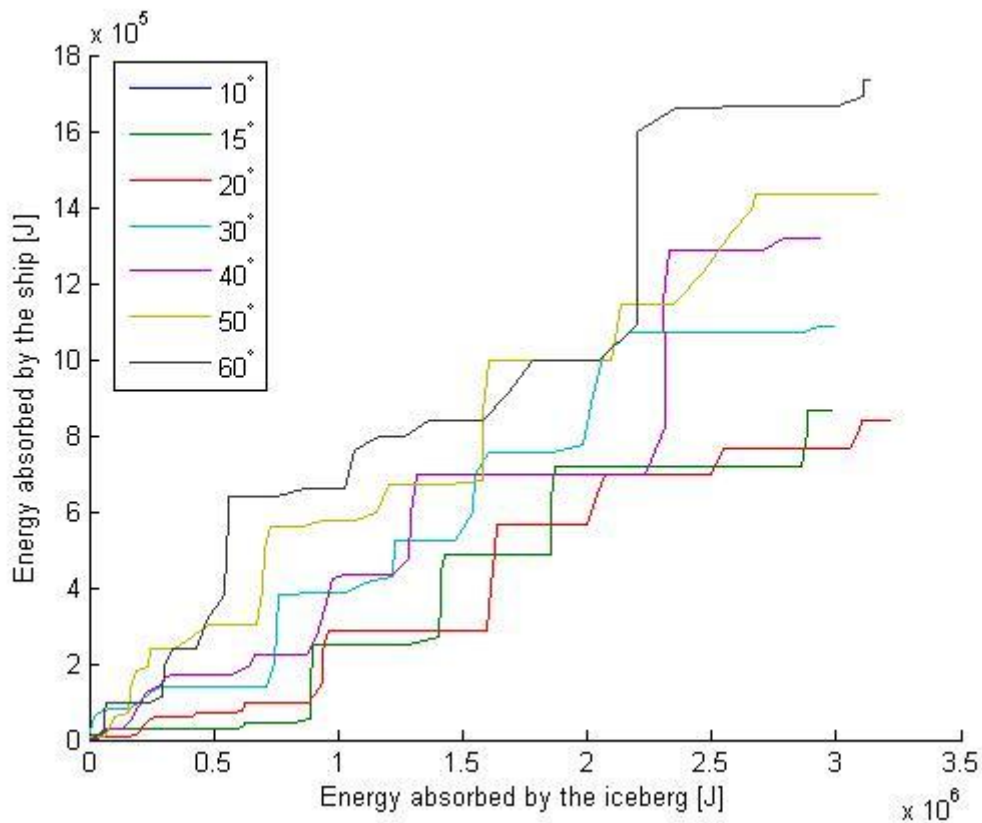
Figure 39: Distribution of energy between ship and iceberg for different cone angles on the ship without ice strengthening. Note the difference in the scale between the x-axis and the y-axis.

In very early stages the 30° iceberg does most damage to the ship, but the 50° and then the 60° iceberg soon takes over. After 0.5m deformation the 60° iceberg has given over half as much damage as it has absorbed.
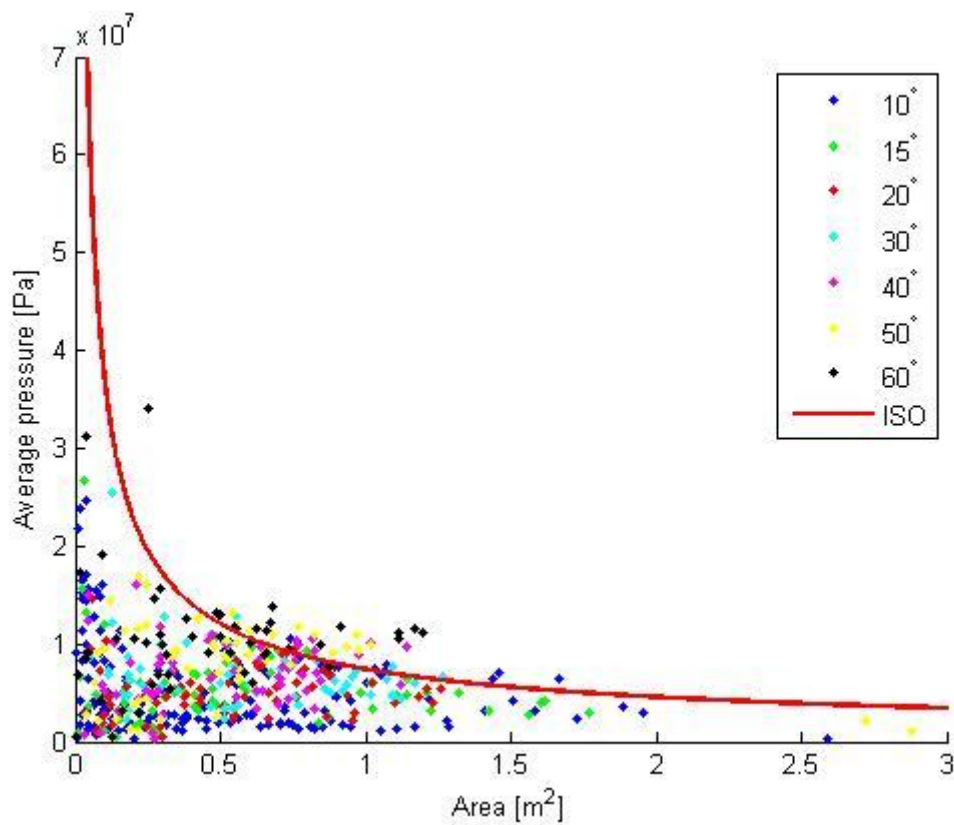
Figure 40: Average pressure-nominal area relationship different cone angles on the ship without ice strengthening

In Figure 40 there is seen that the pressures were increasing for higher angle of the cone. For the highest angles the pressure was well above the pressure curve from the ISO-standard [5].
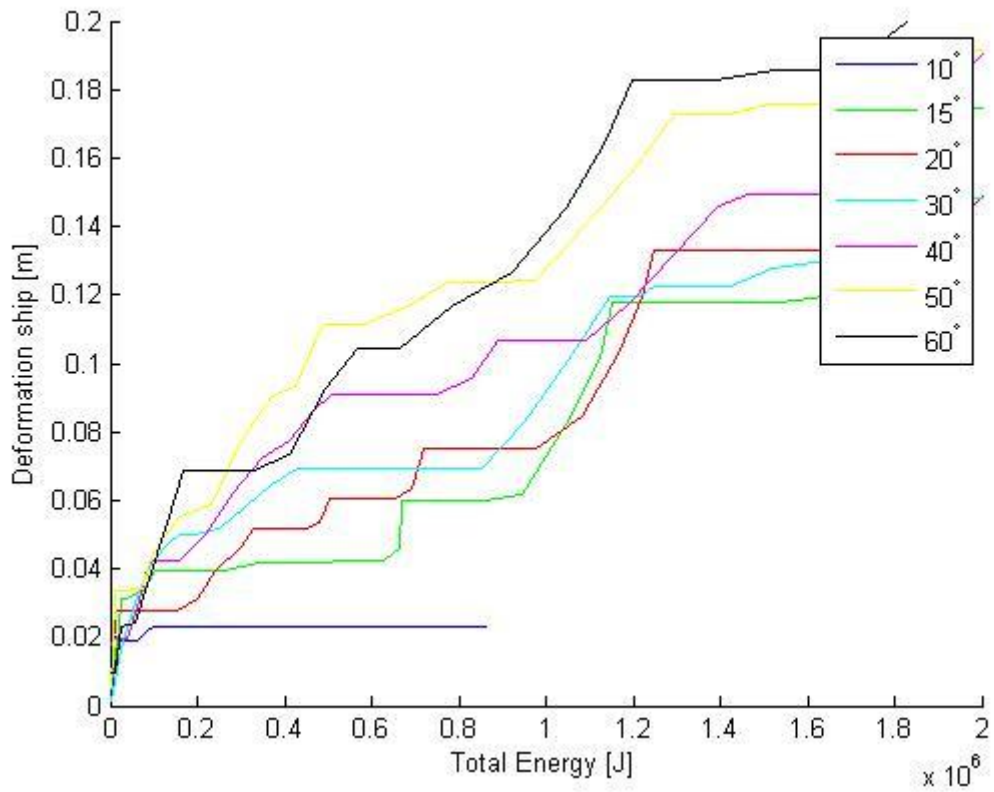
Figure 41: Total collision energy - deformation of shipside for the ship without ice strengthening for different cone angles

As seen in Figure 41 the deformation rise quickly with a cone angle higher than used in the first part of this analysis. The increase in deformation between the steps in cone angle then becomes smaller for each higher step and the 50° and 60° gave about equal in deformation.

3.1.5 Angle check for the ship with 1C ice strengthening
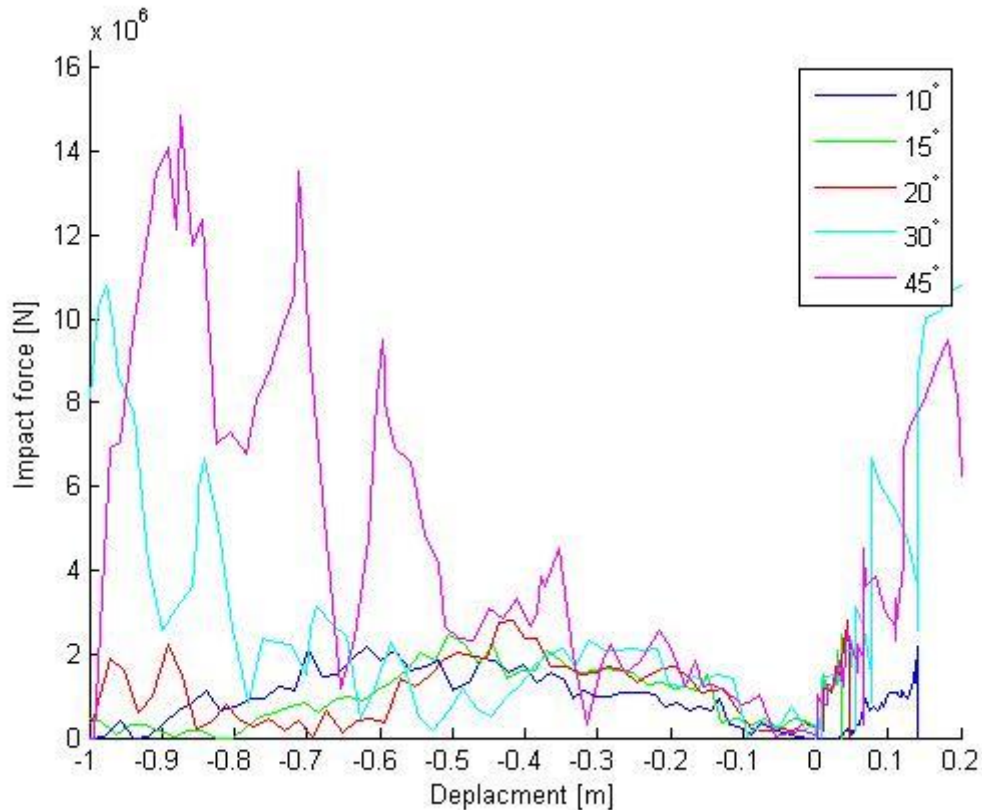
Figure 42: Impact force against displacement for different cone angles on the ship with 1C ice strengthening. Ship to the right and iceberg to the left.

The result in Figure 42 shows the impact force – displacement curve to check how the angle of the cone influences the confinement. The force from the 30° and 45° iceberg continues to rise with higher deformation, but the area increase much faster with a high angle so the pressure was about constant. The forces for the 45° iceberg and at high deformations the 30° iceberg, shows the same behavior as the largest icebergs from Figure 31 and Figure 38. The force builds up and the ice fractures in shear and then the forces build up again for another fracture. The deformation of the iceberg at the peak of the impact force decrease with increased angle of the cone. However as in Figure 38, the impact force and the damage to the ship until the peak are increasing with increasing angle. This increase in force may affect the confinement necessary to keep the iceberg from braking and thereby cause a fall in the impact force at a smaller deformation.

54
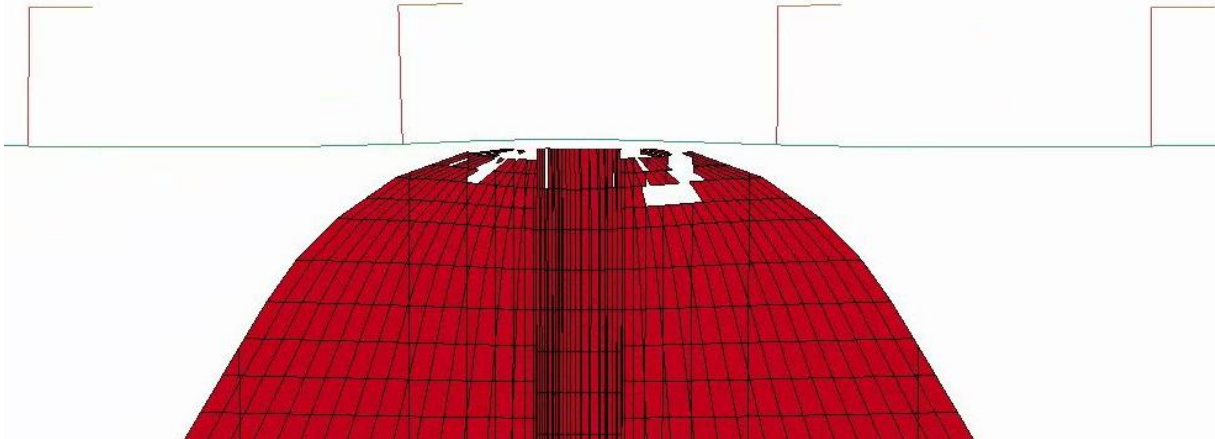
Figure 43: Deformation of 30° iceberg on the 1C ship

Seen in Figure 43 the shear fracture was beginning at an earlier stage than in Figure 19. This fracture was happening at the same impact force for each of the iceberg, but at different deformation.

As seen in Figure 38 and Figure 42 the deflection of the shipside was increasing with higher cone angle. The different cone angles are following the same force deformation curve for the shipside independent of the cone angle. There was some exceptions the 10° iceberg were from about 0.05 m of deflection on the 1C ice strengthened ship in Figure 42 needing less force to deform the shipside. This was an error in the simulations as described below Figure 28. Since the other icebergs was following about the same curve the both for the ship without ice strengthening and for the 1C ice strengthened ship, the strength of the shipside seems to not be affected by the change in cone angle.

The impact force-deformation curve on the iceberg side in Figure 38 and Figure 42 was altered between the different cone angles and the force was built up faster with higher cone angles causing the iceberg to be stiffer. This can be seen in Figure 39 and Figure 44 as a higher gradient because the faster build up in impact force on the iceberg side give less deformation and thereby less energy was absorbed in the ice.
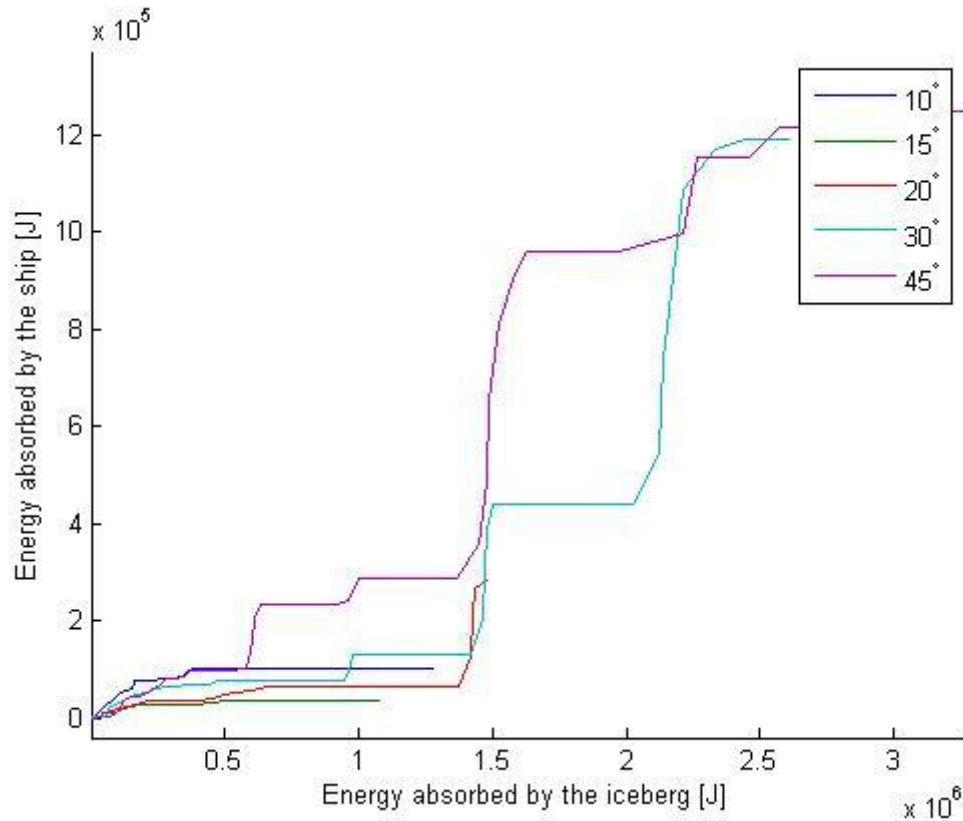
Figure 44: Distribution of energy between ship and iceberg for different cone angles on the ship with 1C ice strengthening. Note the difference in the scale between the x-axis and the y-axis.

As in the Figure 39 the gradient are increasing with higher cone angle when not considering 10° cone angle iceberg. All icebergs do about equal damage to the ship when the iceberg has absorbed less than $3 \cdot 10^5$ joule of energy. From this point on the 45° iceberg have the steepest gradient, but the 30° iceberg does nearly as much damage in the end.
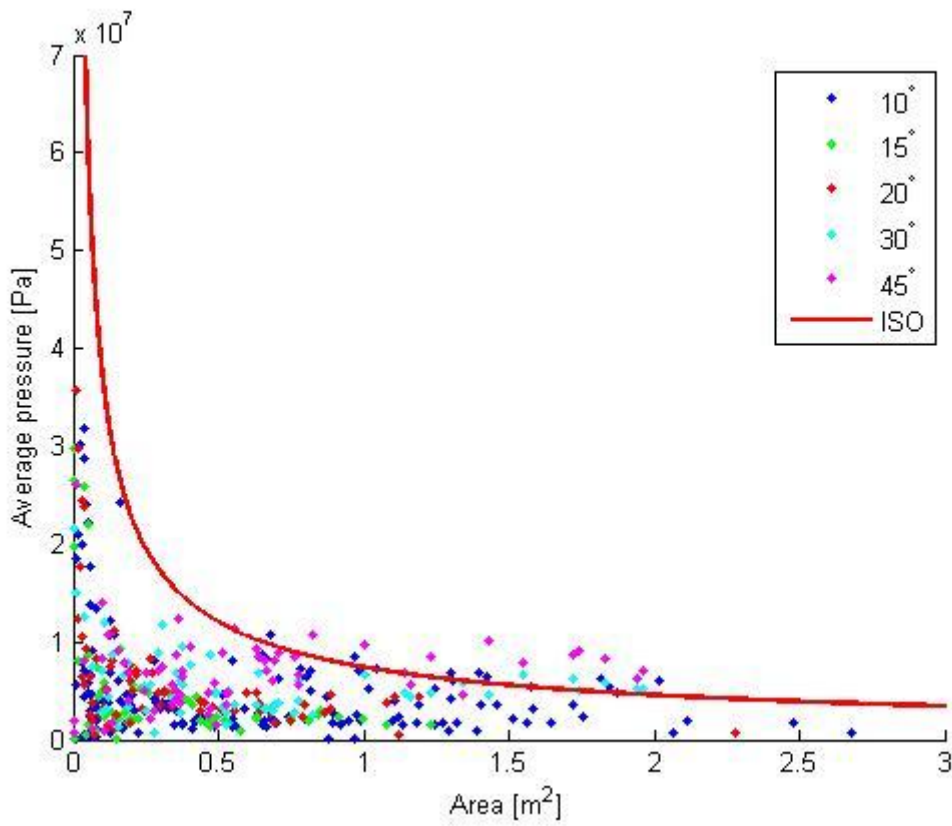
Figure 45: Average pressure-nominal area relationship different cone angles on the ship with 1C ice strengthening

The pressure was rising as the angle were higher as with the ship without ice strengthening the 45° Iceberg gave pressures much higher than the curve in the ISO standard [5].
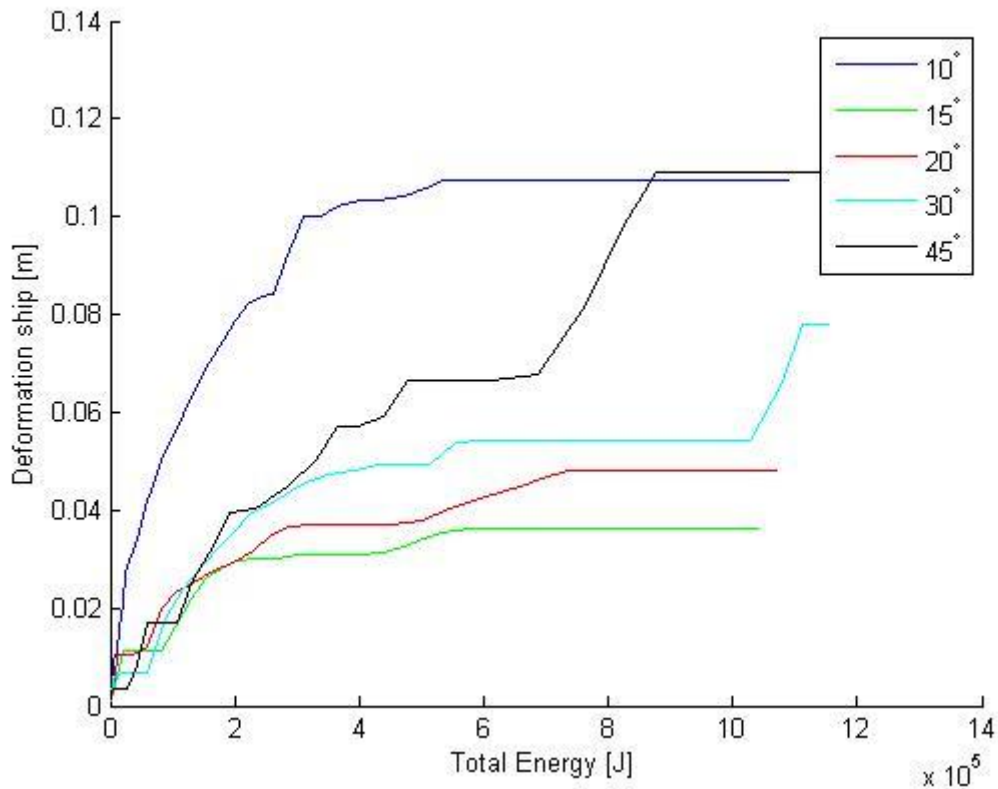
Figure 46: Total collision energy - deformation of shipside for the ship with 1C ice strengthening for different cone angles

Figure 46 show that the 10° iceberg deforms the ship with little collision energy. This was caused by that the elements in the middle was going through the shipside without rupturing the shipside Figure 28. The other icebergs the tendency was that the deformation at a given energy was higher with higher cone angle.

4   Conclusion

In this thesis there were done numerical simulations of different iceberg shapes against three different hulls with different ice classes. The objective was to evaluate different iceberg shapes in terms of damage to the shipside.

Only the contact zone of the iceberg was modeled (chapter 2.1.3) as well as a section of the shipside were modeled (chapter 2.2.2). This was done to shorten the time to solve the analysis, and because only a part of the iceberg will hit the shipside.

None of the icebergs was able to rupture the shipsides plating and there were difficult to find a shape that give more damage to all of the shipsided that was tested. The 1.68 m in radius iceberg gave the most energy to the ship relative to energy absorbed in the iceberg for two of the tested shipsides. Because of that the result of the smallest iceberg on the 1A ice strengthened ship was uncertain the 1.68 m in radius iceberg possibly also gives the highest energy to the ship relative to it self. Therefore it seems to be more onerous for the shipside with a larger contact area with the iceberg. Since there were not tested any larger icebergs there can not be concluded with more than that the iceberg with a radius of 1.68 m in front were the most onerous in this analysis.

The analysis of the cone angle was more conclusive and the iceberg became stronger with higher angle of the cone, but seemed to flatten out at about 50°.

As this thesis only has tested a few different combinations of iceberg shape and shipside scantlings, there should be done further research with the combinations of shipside scantling and iceberg shapes. The angle of the cone and radius of the tip should be combined in more combinations to see if the result still is the same.

Since the largest iceberg did most damage there also should be tested larger radius in front of the iceberg to see if there can be done more damage with a larger iceberg.

There should also be tested with smaller element in the iceberg for the smallest icebergs as these gave result that coincided with the element length.

5 <u>Acknowledge</u>

The references have been referenced by the ACM system. Thanks to Jørgen Amdalh, Martin Storheim and Ekaterina Kim for the support during the work on this thesis.

Bibliography

1   NORSOK. *NORSOK N-001*. 2004.

2   NORSOK. *NORSOK N-004*. 2004.

3   NORSOK. *NORSOK N-003*. 2007.

4   Liu, Zhenhui and Amdahl, Jørgen. A new formulation of the impact mechanics of ship collisions and its application to a ship-iceberg collision. *Marine Structures* (2010), 360-384.

5   *ISO/CD 19906*. Geneva, 2010.

6   Moan, Torgeir. *TMR 4190 Finite element modelling and analysis of marine structures*. Department of marine technology, Norwegian university of science and technology, Trondheim, 2003.

7   Gagnon, Robert E. Results of numerical simulations of growler impact tests. *Cold Regions Science and Technology* (2007), 206-214.

8   Gagnon, Robert E. Numerical simulations of a tanker collision with a bergy bit incorporating hydrodynamics, a validated ice model and damage to the.. *Cold Reg. Sci. Technol.* (2012).

9   Liu, Zhenhui, Amdahl, Jørgen, and Løset, Sveinung. Plasticity based material modelling of ice and its application to ship–iceberg impacts (2010).

10 Liu, Zhenhui, Amdahl, Jørgen, and Løset, Sveinung. Integrated numerical analysis of an iceberg collision with a foreship structure (2011).

11 Liu, Zhenhui, Amdahl, Jørgen, and løset, Sveinung. Integrated numerical analysis of iceberg collision with ship side structures. In *International Conference on Port and Ocean Engineering* (Montreal 2011).

12 Storheim, Martin, Kim, Ekaterina, Amdahl, Jørgen, and Ehlers, Søren. ICEBERG SHAPE
SENSITIVITY IN SHIP IMPACT ASSESSMENT IN VIEW OF. In *Proceedings of The 31st
International Conference on Ocean* (Rio de Janeiro 2012), Offshore and Arctic
Engineering.

13 Pedersen, Leif Toudal and Jensen, Torben P. *EMU | Danmarks Undervisningsportal*. 2007.

14 Whitworth, Robert W. and Petrenko, Victor F. *Physics of ice*. Oxford University press,
1999.

15 Derradji Aouat, Asmed. Multi-surface failure criterion for saline ice in the brittle regime
(2003).

16 Alsos, Hagbart Skage. On the resistance to penetration of stiffened plates. *International
Journal of Impact Engineering* (2009), 875-887.

17 Betytschko, T and Tsay, C S. Explicit algorithms for the nonlinear dynamics of shells.
Comput. Methods Appl. Mech. Engrg (1984), 225-251.

18 Meek, J L. *Cumputer methods in structural analysis*. E & FN SPON, Singapore, 1991.

19 Daley, C G. Reanalysis of ice pressure-area. *Marine Technology and SNAME News* (2007),
234.

20 Alsos, Hagbart S. Analytical and numerical analysis of sheet metal instability using a stress
based criterion. *International Journal of Solids and Structures* (2008), 2042-2055.

21 Tangen, Lisbeth. *Referanseliste i APA-stil*.

22 Veritas, Det Norske. *DNV-RP-C204: Design against Accidental Loads*. 2010.

23 Daley, Claude and Kim, Hyunwook. *ICE COLLISION FORCES CONSIDERING STRUCTURAL
DEFORMATION*. Memorial University of Newfoundland, St. John's, Newfoundland and
Labrador, Canada, 2010.

24 Gagnon, R E. *A numerical model of ice crushing using a foam analogue*. Cold Regions

Science and Technology, Canada, 2011.

25 Suh, Yongsuk, Jang, Kibok, Ito, Hisashi et al. *DEVELOPMENT OF ICE CLASS ARC4 MK III*.

Proceedings of the ASME 27th International Conference on Offshore Mechanics and

Arctic Engineering, 2008.

Appendix: Matlab
Script for drawing and
running the analysis

Norwegian University of Science and Technology
Department of Marine Technology

A. Appendix: Matlab Script for drawing and running the analysis

```matlab
%Input values for the shipside
stiveravstand = 0.8;
stiveriavstand = 0.8;
Indrestiverstegtykkelse = 0.013;
Indrestiverflenstykkelse = 0.0384;
stiveruhoyde = 0.37;
stiverubredde = 0.0535;
ytrestiverstegtykkelse = 0.013;
ytrestiverflenstykkelse = 0.0384;
stiveribredde = 0.0535;
stiverihoyde = 0.37;
stringeravstand = stiveravstand*10;
rammeavstand = 3.20;
antallrammer = 6;
antallstringer = 2;
Stringertykkelse = 0.012;
Girdertykkelse = 0.0155;
avstandindreytre = 3.6;
ytrehudtykkelse = 0.017;
indrehudtykkelse = 0.015;
Girderstivertykkelse = 0.0115;
rammestiverhoyde = 0.25;
Girderstiverflenstykkelse = 0.015;
Girderstiverflenbredde = 0.1;

filnavnIS = 'iceberg';

%Input values for the iceberg
Meshsize = 0.1;
H = 0.7;
%Used varius angles
Angle = [15, 40];
%Used varius radiuses
Radius =[(stiveravstand/4); (stiveravstand/2); (stiveravstand); (rammeavstand/4); (rammeavstand)];
Radiustext ={num2str(stiveravstand/4); num2str(stiveravstand/2); num2str(stiveravstand);
num2str(rammeavstand/4); num2str(rammeavstand)};

%For loops to go through all varius input values
for iii=1
    for kkk=1:3
        %Folder for the analysis
        cd 'C:\Users\Torstein\Documents\utenis'
        %Give different name for each model
        filnavn = num2str(Angle(iii));

        %Calculate mesh size based on the thickness of the plating and avoiding
        %transitions
        Meshsizelangships=rammeavstand/floor((rammeavstand)/(min([indrehudtykkelse,ytrehudtykkelse,...
        Stringertykkelse,ytrestiverflenstykkelse,ytrestiverstegtykkelse,Indrestiverflenstykkelse,...
        Indrestiverstegtykkelse])*10));
        Meshsizetverrships = stiveravstand/floor(stiveravstand/(min([indrehudtykkelse,ytrehudtykkelse,...
        Girdertykkelse])*10));
        Meshsizeouterstif = min([stiveruhoyde/3, ytrestiverstegtykkelse*10]);
        Meshsizeinnerstif = min([stiverihoyde/3, Indrestiverstegtykkelse*10]);
        Meshsizetverrstifffl = ytrestiverflenstykkelse*10;
        Meshsizetverrstiff = min([ytrestiverstegtykkelse*10,stiveruhoyde/3]);
        Meshsizetverrstringer = min([Stringertykkelse*10 , avstandindreytre/3]);
        Meshsizetverrgirder = min([Girdertykkelse*10 , avstandindreytre/3]);

        %Begins the controll file for Patran
        File = {'$# Recorded by: Patran 2012 64-Bit';...
            'uil_file_new.go( "C:\MSC.Software\Patran_x64\20121/template.db",  @';...
            %Specify filename and location
```

Appendix: Matlab Script for drawing and running the analysis

```matlab
    ['"C:\Users\Torstein\Documents\utenis\' filnavn '.db" )'];...
    '$# Question from application FILE';...
    %If already exiting overwrite
    ['$#     Database C:\Users\Torstein\Documents\utenis\' filnavn '.db already exists.'];...
    '$# Do you wish to delete the existing database and create a new one ?';...
    '$? YES 36000002';...
    '$# Copying C:\MSC.Software\Patran_x64\20121/template.db to';...
    ['$# C:\Users\Torstein\Documents\utenis\' filnavn '.db'];...
    '$# Template copy complete.';...
    '$# Database version 3.12 created by Patran 2012 64-Bit successfully opened.';...
    ['$# Creating journal file C:\Users\Torstein\Documents\utenis\' filnavn '.db.jou at'];...
    '$# 28-Jan-13 17:37:19';...
    '$# Synchronizing the User Data Manager';...
    'set_current_dir( "C:\Users\Torstein\Documents\utenis" )';...
    '$# The current working directory has been changed to the database location';...
    '$# shown.';...
    '$# ';...
    '$#';...
    %Specify LS-DYNA as solver
    'uil_pref_analysis.set_analysis_preference( "LS-DYNA3D", "Structural", ".key",  @';...
    '"", "No Mapping" )';...
    '$#  Loading... ls_dyna3d_defn_dflt_tbs( LS-DYNA3D, Structural ));';...
    %Make a group for the outer hull
    'ga_group_create( "outerhull" )';...
    'ga_group_current_set( "outerhull" )';...
    '$# Group "outerhull" created.';...
    %Add steel to the material list
    'material.create( "Analysis code ID", 10001, "Analysis type ID", 1, "Steel", 1, @';...
    ' "", "Isotropic", 1, "Directionality", 1, "Linearity", 1, "Homogeneous", 0,  @';...
    '"Linear Elastic", 1, "Model Options & IDs", ["Linear Elastic(MAT1)", "Solid",  @';...
    '"", "", ""], [11002, 11003, 0, 0, 0], "Active Flag", 1, "Create", 11,  @';...
    '"External Flag", FALSE, "Property IDs", ["Density", "Elastic Modulus",  @';...
    '"Poisson Ratio"], [16, 2, 5, 0], "Property Values", ["7800", "2.1E11", "0.3",  @';...
    '""] )';...
    'STRING asm_create_line_xyz_created_ids[VIRTUAL]'};

%Counting lines in control file
k = 33;

%Draw one curve for the plate between each stiffener
for i=1:stringeravstand*(antallstringer+1)/stiveravstand
File(i*3-2+k:i*3+k) = {['asm_const_line_xyz( "' num2str(i) '", "<' num2str(stiveravstand)...
' 0 0>", "[' num2str((i-1)*stiveravstand) ' 0 0]", "Coord 0",  @'];...
        'asm_create_line_xyz_created_ids )';...
        ['$# 1 Line created: Line ' num2str(i)]};
end

%Count number of curves drawn
Curve = stringeravstand*(antallstringer+1)/stiveravstand;

k = k+stringeravstand*(antallstringer+1)*3/stiveravstand;

%Extrude the curves to surfaces
File(1+k:k+4) = {'STRING sgm_sweep_surface_e_created_ids[VIRTUAL]';...
    ['sgm_const_surface_extrude( "1", "<0 0 ' num2str(rammeavstand*(antallrammer+1)) '>", 1., 0., "[0
0 0]", "Coord 0",  @'];...
    ['"Curve 1:' num2str(Curve) '", sgm_sweep_surface_e_created_ids )'];...
    ['$# 11 Surfaces Created: Surfaces 1:' num2str(Curve)]};

k = k+4;

%Count surfaces made
Surface = Curve;

%Place mesh seed on the edges of the plates
for i = 1:Surface

File(k+1:k+4)={['mesh_seed_create( "Surface ' num2str(i) '.3", 2, 0, 0., '...
num2str(Meshsizelangships) ', 0. )'];...
['mesh_seed_create( "Surface ' num2str(i) '.2", 2, 0, 0., ' num2str(Meshsizetverrships) ', 0. )'];...
['mesh_seed_create( "Surface ' num2str(i) '.4", 2, 0, 0., ' num2str(Meshsizetverrships) ', 0. )'];...
['mesh_seed_create( "Surface ' num2str(i) '.1", 2, 0, 0., ' num2str(Meshsizelangships) ', 0. )']};

k = k+4;
```

NTNU
Norwegian University of Science and Technology
Department of Marine Technology

Appendix: Matlab
Script for drawing and
running the analysis

```matlab
    end

%Add the properties to the surfaces made
File(k+1:k+3)={'elementprops_create( "Outherhull", 51, 25, 35, 11003, 1, 20, [13, 20, 36, 1004, @';...
    ' 11044, 11136, 11026, 11027, 1011, 11182], [5, 2, 1, 1, 4, 3, 4, 4, 1, 1], [ @';...
    ['"m:Steel", "", "' num2str(ytrehudtykkelse) '", "", "", "", "", "", "", ""],...
        "Surface 1:' num2str(Surface) ':1 " )']};


k = k+3;

%Mesh the surfaces and add a new group for the outer stiffeners webs
File(k+1:k+14) = {'ui_exec_function( "mesh_seed_display_mgr", "init" )';...
    'INTEGER fem_create_mesh_surfa_num_nodes';...
    'INTEGER fem_create_mesh_surfa_num_elems';...
    'STRING fem_create_mesh_s_nodes_created[VIRTUAL]';...
    'STRING fem_create_mesh_s_elems_created[VIRTUAL]';...
    'fem_create_mesh_surf_4( "IsoMesh", 49152,  @';...
    ['"Surface 1:' num2str(Surface)...
    '", 1, ["0.2"], "Quad4", "#", "#", "Coord 0", "Coord 0",  @'];...
    'fem_create_mesh_surfa_num_nodes, fem_create_mesh_surfa_num_elems,  @';...
    'fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )';...
    'ga_group_create( "outerstiffenerweb" )';...
    'ga_group_current_set( "outerstiffenerweb" )';...
    '$# Group "outerstiffenerweb" created.';...
    'STRING asm_create_line_xyz_created_ids[VIRTUAL]';...
    'ui_exec_function( "mesh_seed_display_mgr", "init" )'};

k = k+14;

%Add boundary conditions to the outer hull plating
File(k+1:k+4) = {'loadsbcs_create2( "BCouterhull", "Displacement", "Nodal", "", "Static", [ @';...
    ['"Surface 1.1 ' num2str(Surface) '.3 ' num2str(1)  ':' num2str(Surface) '.2 '...
    num2str(1) ':' num2str(Surface) '.4 '...
    '"], "Geometry", "Coord 0", "1.", ["<0 0 0>",  @'];...
    '"<0 0 0>", "<    >", "<    >"], ["", "", "", ""] )';...
    '$# Load/BC set "BC1" created.'};

k = k+4;

%Draw curves for the stiffener web and exclude the points were there are stringers
for j = 0:antallstringer
    for i = 1+j*stringeravstand/stiveravstand:stringeravstand/stiveravstand...
            +j*stringeravstand/stiveravstand-1
        File((i-j)*3-2+k:(i-j)*3+k) = {['asm_const_line_xyz( "' num2str(i+Curve-j)...
            '", "<0 ' num2str(stiveruhoyde) ' 0>", "[' num2str(i*stiveravstand)...
            ' 0 0]", "Coord 0",  @'];...
            'asm_create_line_xyz_created_ids )';...
            ['$# 1 Line created: Line ' num2str(i+Curve-j)]};
    end
end

%Add a new variable giving the number of the latest surface to be able
%to know the number of the first and last surface that is the stiffeners web
Surfacenew = Surface + stringeravstand*(antallstringer+1)/stiveravstand - 1-antallstringer;
Curvenew = Curve + stringeravstand*(antallstringer+1)/stiveravstand - 1-antallstringer;
k = k+((antallstringer+1)*stringeravstand/stiveravstand-1-antallstringer)*3;


%Make surfaces of the curves
File(1+k:k+4) = {'STRING sgm_sweep_surface_e_created_ids[VIRTUAL]';...
    ['sgm_const_surface_extrude( "' num2str(Surface+1) '", "<0 0 ' ...
    num2str(rammeavstand*(antallrammer+1)) '>", 1., 0., "[0 0 0]", "Coord 0",  @'];...
    ['"Curve ' num2str(Curve+1) ':' num2str(Curvenew) '", sgm_sweep_surface_e_created_ids )'];...
    ['$# ' num2str(Surfacenew-Surface) ' Surfaces Created: Surfaces '...
    num2str(Surface+1) ':' num2str(Surfacenew)]};


Firstsurface = Surface;
Surface = Surfacenew;
Surfaceend = Surface;
Curve = Curvenew;
k = k+4;
```

Appendix: Matlab Script for drawing and running the analysis

```matlab
%Add properties to the surfaces outer stiffener web
File(k+1:k+3)={'elementprops_create( "outerstiffenerweb", 51, 25, 35, 11003, 1, 20, [13, 20, 36, 1004,
@';...
    ' 11044, 11136, 11026, 11027, 1011, 11182], [5, 2, 1, 1, 4, 3, 4, 4, 1, 1], [ @';...
    ['"m:Steel", "", "' num2str(ytrestiverstegtykkelse) '", "", "", "", "", "", "", ""], "Surface '
num2str(Firstsurface+1) ':' num2str(Surface) ':1 '...
    '" )']};


k = k+3;

%Add mesh seed to the edges
for i = Firstsurface+1:Surface
    File(k+1:k+4)={['mesh_seed_create( "Surface ' num2str(i) '.3", 2, 0, 0., '
num2str(Meshsizelangships) ', 0. )'];...
        ['mesh_seed_create( "Surface ' num2str(i) '.2", 2, 0, 0., ' num2str(Meshsizetverrstiff) ',
0. )'];...
        ['mesh_seed_create( "Surface ' num2str(i) '.4", 2, 0, 0., ' num2str(Meshsizetverrstiff) ',
0. )'];...
        ['mesh_seed_create( "Surface ' num2str(i) '.1", 2, 0, 0., ' num2str(Meshsizelangships) ',
0. )']};
    k = k+4;
end


%Mesh surfaces and make a group for the outer stiffeners flange
File(k+1:k+9) = {'fem_create_mesh_surf_4( "IsoMesh", 49152,  @';...
    ['"Surface ' num2str(Firstsurface+1) ':' num2str(Surface) ':1 '...
    '" @'];...
    ', 1, ["0.2"], "Quad4", "#", "#", "Coord 0", "Coord 0",  @';...
    'fem_create_mesh_surfa_num_nodes, fem_create_mesh_surfa_num_elems,  @';...
    'fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )';...
    'ga_group_create( "outerstiffenerflange" )';...
    'ga_group_current_set( "outerstiffenerflange" )';...
    '$# Group "outerstiffenerflange" created.';...
    'STRING asm_create_line_xyz_created_ids[VIRTUAL]'};

k = k+9;

%Add boundary conditions
File(k+1:k+4) = {'loadsbcs_create2( "BCstiffenerweb", "Displacement", "Nodal", "", "Static", [ @';...
    ['"Surface ' num2str(Firstsurface+1)  ':' num2str(Surface) '.2 ' num2str(Firstsurface+1) ':'
num2str(Surface) '.4'...
    '"], "Geometry", "Coord 0", "1.", ["<0 0 0>",  @'];...
    '"<0 0 0>", "<      >", "<      >"], ["", "", "", ""] )';...
    '$# Load/BC set "BC1" created.'};

k = k+4;

%Draw lines for the flange and exclude the points were there are stringers
for j = 0:antallstringer
    for i =
1+j*stringeravstand/stiveravstand:stringeravstand/stiveravstand+j*stringeravstand/stiveravstand-1
        File((i-j)*3-2+k:(i-j)*3+k) = {['asm_const_line_xyz( "' num2str(i+Curve-j) '", "<'
num2str(stiverubredde) ' 0 0>", "[' num2str(i*stiveravstand) ' ' num2str(stiveruhoyde) ' 0]", "Coord 0",
@'];...
        'asm_create_line_xyz_created_ids )';...
        ['$# 1 Line created: Line ' num2str(i+Curve-j)]};
    end
end


Surfacenew = Surface + stringeravstand*(antallstringer+1)/stiveravstand - 1-antallstringer;
Curvenew = Curve + stringeravstand*(antallstringer+1)/stiveravstand - 1-antallstringer;
k = k+((antallstringer+1)*stringeravstand/stiveravstand-1-antallstringer)*3;

%Extrude curves to surfaces
File(1+k:k+4) = {'STRING sgm_sweep_surface_e_created_ids[VIRTUAL]';...
    ['sgm_const_surface_extrude( "' num2str(Surface+1) '", "<0 0 '
num2str(rammeavstand*(antallrammer+1)) '>", 1., 0., "[0 0 0]", "Coord 0",  @'];...
    ['"Curve ' num2str(Curve+1) ':' num2str(Curvenew) '", sgm_sweep_surface_e_created_ids )'];...
```

NTNU
Norwegian University of Science and Technology
Department of Marine Technology

Appendix: Matlab
Script for drawing and
running the analysis

```matlab
            ['$# ' num2str(Surfacenew-Surface) ' Surfaces Created: Surfaces ' num2str(Surface+1) ':'
num2str(Surfacenew)]};


        Firstsurface = Surface;
        Surface = Surfacenew;
        Surfaceend = Surface;
        Curve = Curvenew;
        k = k+4;


        %Add mesh seed to the edges
        for i = Firstsurface+1:Surface
            File(k+1:k+4)={['mesh_seed_create( "Surface ' num2str(i) '.3", 2, 0, 0., '
num2str(Meshsizelangships) ', 0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.2", 2, 0, 0., ' num2str(Meshsizetverrstiff) ',
0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.4", 2, 0, 0., ' num2str(Meshsizetverrstiff) ',
0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.1", 2, 0, 0., ' num2str(Meshsizelangships) ',
0. )']};
            k = k+4;
        end

        %Add properties to the surfaces
        File(k+1:k+3)={'elementprops_create( "outerstiffenerflange", 51, 25, 35, 11003, 1, 20, [13, 20, 36,
1004, @';...
                ' 11044, 11136, 11026, 11027, 1011, 11182], [5, 2, 1, 1, 4, 3, 4, 4, 1, 1], [ @';...
                ['"m:Steel", "", "' num2str(ytrestiverflenstykkelse) '", "", "", "", "", "", "", ""], "Surface '
num2str(Firstsurface+1) ':' num2str(Surface) ':1 '...
                '" )']};


        k = k+3;

        %Mesh and create group for the Stringers
        File(k+1:k+9) = {'fem_create_mesh_surf_4( "IsoMesh", 49152,  @';...
            ['"Surface ' num2str(Firstsurface+1) ':' num2str(Surface) ':1 '...
            '" @'];...
            ', 1, ["0.2"], "Quad4", "#", "#", "Coord 0", "Coord 0",  @';...
            'fem_create_mesh_surfa_num_nodes, fem_create_mesh_surfa_num_elems,  @';...
            'fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )';...
            'ga_group_create( "Stringer" )';...
            'ga_group_current_set( "Stringer" )';...
            '$# Group "Stringer" created.';...
            'STRING asm_create_line_xyz_created_ids[VIRTUAL]'};

        k = k+9;
        %Create boundary conditions
        File(k+1:k+4) = {'loadsbcs_create2( "BCstiffenerflange", "Displacement", "Nodal", "", "Static",
[ @';...
            ['"Surface ' num2str(Firstsurface+1)  ':' num2str(Surface) '.2 ' num2str(Firstsurface+1) ':'
num2str(Surface) '.4'...
            '"], "Geometry", "Coord 0", "1.", ["<0 0 0>",  @';...
            '"<0 0 0>", "<      >", "<      >"], ["", "", "", ""] )';...
            '$# Load/BC set "BC1" created.'};

        k = k+4;

        %Draw the curves for the stringers
        for i=1:antallstringer
            File(i*3-2+k:i*3+k) = {['asm_const_line_xyz( "' num2str(i+Curve) '", "<0 '
num2str(avstandindreytre) ' 0>", "[' num2str(i*stringeravstand) ' 0 0]", "Coord 0",  @';...
                'asm_create_line_xyz_created_ids )';...
                ['$# 1 Line created: Line ' num2str(i+Curve)]};
        end

        Curvenew = Curve + antallstringer;
        Surfacenew = Surface + antallstringer;
        k = k+antallstringer*3;

        %Make surface from the curves
        File(1+k:k+4) = {'STRING sgm_sweep_surface_e_created_ids[VIRTUAL]';...
```

```matlab
                ['sgm_const_surface_extrude( "' num2str(Surface+1) '", "<0 0 '
num2str(rammeavstand*(antallrammer+1)) '>", 1., 0., "[0 0 0]", "Coord 0",  @'];...
                ['"Curve ' num2str(Curve+1) ':' num2str(Curvenew) '", sgm_sweep_surface_e_created_ids )'];...
                ['$# ' num2str(Curvenew-Curve) ' Surfaces Created: Surfaces ' num2str(Surface+1) ':'
num2str(Surfacenew)]};

        k = k+4;
        Firstsurface = Surface;
        Surface = Surfacenew;
        Surfaceend = Surface;

        %Create mesh seed on the edges
        for i = Firstsurface+1:Surface
            File(k+1:k+4)={['mesh_seed_create( "Surface ' num2str(i) '.3", 2, 0, 0., '
num2str(Meshsizelangships) ', 0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.2", 2, 0, 0., ' num2str(Meshsizetverrstringer) ',
0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.4", 2, 0, 0., ' num2str(Meshsizetverrstringer) ',
0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.1", 2, 0, 0., ' num2str(Meshsizelangships) ',
0. )']};
            k = k+4;
        end


        %Add properties to the stringers
        File(k+1:k+3) = {'elementprops_create( "Stringer", 51, 25, 35, 11003, 1, 20, [13, 20, 36, 1004, @';...
            ' 11044, 11136, 11026, 11027, 1011, 11182], [5, 2, 1, 1, 4, 3, 4, 4, 1, 1], [ @';...
            ['"m:Steel", "", "' num2str(Stringertykkelse) '", "", "", "", "", "", "", ""], "Surface '
num2str(Firstsurface+1) ':' num2str(Surface) '" )']};

        k = k+3;

        %Mesh the stringers
        File(k+1:k+5) = {'fem_create_mesh_surf_4( "IsoMesh", 49152,  @';...
            ['"Surface '  num2str(Firstsurface+1) ':' num2str(Surface) '" @'];...
            ', 1, ["0.2"], "Quad4", "#", "#", "Coord 0", "Coord 0",  @';...
            'fem_create_mesh_surfa_num_nodes, fem_create_mesh_surfa_num_elems,  @';...
            'fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )'};
        k = k+5;


        %Add boundary conditions
        File(k+1:k+4) = {'loadsbcs_create2( "BCstringer", "Displacement", "Nodal", "", "Static", [ @';...
            ['"Surface ' num2str(Firstsurface+1) ':' num2str(Surface) '.2 ' num2str(Firstsurface+1) ':'
num2str(Surface) '.4"], "Geometry", "Coord 0", "1.", ["<0 0 0>",  @'];...
            '"<0 0 0>", "<      >", "<      >"], ["", "", "", ""] )';...
            '$# Load/BC set "BC1" created.'};

        k = k+4;


        %Make group for the web frames
        File(k+1:k+4) = {'ga_group_create( "Girder" )';...
            'ga_group_current_set( "Girder" )';...
            '$# Group "Girder" created.';...
            'STRING asm_create_line_xyz_created_ids[VIRTUAL]'};

        Curve = Curvenew;
        k = k+4;

        %Draw curves for the web frames
        for i=1:antallrammer
            File(i*3-2+k:i*3+k) = {['asm_const_line_xyz( "' num2str(i+Curve) '", "<0 '
num2str(avstandindreytre) ' 0>", "[0 0 ' num2str(i*rammeavstand) ']", "Coord 0",  @'];...
                'asm_create_line_xyz_created_ids )';...
                ['$# 1 Line created: Line ' num2str(i+Curve)]};
        end

        Surfacenew = Surface + antallrammer;
        Curvenew = Curve + antallrammer;
        k = k+antallrammer*3;
```

hi
egian University of Science and Technology
rtment of Marine Technology

Appendix: Matlab
Script for drawing and
running the analysis

```matlab
%Extrude the curves to new surfaces
File(1+k:k+4) = {'STRING sgm_sweep_surface_e_created_ids[VIRTUAL]';...
    ['sgm_const_surface_extrude( "' num2str(Surface+1) '", "<'
num2str(stringeravstand*(antallstringer+1)) ' 0 0>", 1., 0., "[0 0 0]", "Coord 0",  @'];...
    ['"Curve ' num2str(Curve+1) ':' num2str(Curvenew) '", sgm_sweep_surface_e_created_ids )'];...
    ['$# ' num2str(Curvenew-Curve) ' Surfaces Created: Surfaces ' num2str(Surface+1) ':'
num2str(Surfacenew)]};


    k = k+4;
    Firstsurface = Surface;
    Surface = Surfacenew;
    Surfaceend = Surface;
    Curve = Curvenew;


    %Add mesh seed to the edges
    for i = Firstsurface+1:Surface
        File(k+1:k+4)={['mesh_seed_create( "Surface ' num2str(i) '.3", 2, 0, 0., '
num2str(Meshsizetverrships) ', 0. )'];...
            ['mesh_seed_create( "Surface ' num2str(i) '.2", 2, 0, 0., '
num2str(min([Meshsizetverrstringer,Meshsizetverrgirder])) ', 0. )'];...
            ['mesh_seed_create( "Surface ' num2str(i) '.4", 2, 0, 0., '
num2str(min([Meshsizetverrstringer,Meshsizetverrgirder])) ', 0. )'];...
            ['mesh_seed_create( "Surface ' num2str(i) '.1", 2, 0, 0., ' num2str(Meshsizetverrships) ',
0. )']};
        k = k+4;
    end


    %Add properties to the web frames
    File(k+1:k+3) = {'elementprops_create( "Girder", 51, 25, 35, 11003, 1, 20, [13, 20, 36, 1004, @';...
        ' 11044, 11136, 11026, 11027, 1011, 11182], [5, 2, 1, 1, 4, 3, 4, 4, 1, 1], [ @';...
        ['"m:Steel", "", "' num2str(Girdertykkelse) '", "", "", "", "", "", "", ""], "Surface'
num2str(Firstsurface+1) ':' num2str(Surface) '" )']};


    k = k+3;


    %Mesh the web frames
    File(k+1:k+5) = {'fem_create_mesh_surf_4( "IsoMesh", 49152,  @';...
        ['"Surface '  num2str(Firstsurface+1) ':' num2str(Surface) '" @'];...
        ', 1, ["0.2"], "Quad4", "#", "#", "Coord 0", "Coord 0",  @';...
        'fem_create_mesh_surfa_num_nodes, fem_create_mesh_surfa_num_elems,  @';...
        'fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )'};
    k = k+5;

    %Add boundary conditions
    File(k+1:k+4) = {'loadsbcs_create2( "BCgirder", "Displacement", "Nodal", "", "Static", [ @';...
        ['"Surface ' num2str(Firstsurface+1) ':' num2str(Surface) '.2 ' num2str(Firstsurface+1) ':'
num2str(Surface) '.4"], "Geometry", "Coord 0", "1.", ["<0 0 0>",  @'];...
        '"<0 0 0>", "<       >", "<       >"], ["", "", "", ""] )';...
        '$# Load/BC set "BC1" created.'};


    k = k+4;



    %make new group for the inner hull plating
    File(k+1:k+4) ={'ga_group_create( "Innerhull" )';...
        'ga_group_current_set( "Innerhull" )';...
        '$# Group "Innerhull" created.';...
        'STRING asm_create_line_xyz_created_ids[VIRTUAL]'};


    k = k+4;

    %Create curves for the inner hull plating
    for i=1:stringeravstand*(antallstringer+1)/stiveriavstand
        File(i*3-2+k:i*3+k) = {['asm_const_line_xyz( "' num2str(i+Curve) '", "<' num2str(stiveriavstand)
' 0 0>", "[' num2str((i-1)*stiveriavstand) ' ' num2str(avstandindreytre) ' 0]", "Coord 0",  @'];...
            'asm_create_line_xyz_created_ids )';...
            ['$# 1 Line created: Line ' num2str(i+Curve)]};
    end
```

NU
egian University of Science and Technology
artment of Marine Technology

Appendix: Matlab
Script for drawing and
running the analysis

```matlab
        Surfacenew = Surface+stringeravstand*(antallstringer+1)/stiveriavstand;
        Curvenew = Curve+stringeravstand*(antallstringer+1)/stiveriavstand;
        k = k+stringeravstand*(antallstringer+1)*3/stiveriavstand;

        %Extrude the curves to new surfaces
        File(1+k:k+7) = {'STRING sgm_sweep_surface_e_created_ids[VIRTUAL]';...
            ['sgm_const_surface_extrude( "' num2str(Surface+1) '", "<0 0 '
num2str(rammeavstand*(antallrammer+1)) '>", 1., 0., "[0 0 0]", "Coord 0", @'];...
            ['"Curve ' num2str(Curve+1) ':' num2str(Curvenew) '", sgm_sweep_surface_e_created_ids )'];...
            ['$# ' num2str(Curvenew-Curve) ' Surfaces Created: Surfaces ' num2str(Surface+1) ':'
num2str(Surfacenew)];...
            'elementprops_create( "Innerhull", 51, 25, 35, 11003, 1, 20, [13, 20, 36, 1004, @';...
            ' 11044, 11136, 11026, 11027, 1011, 11182], [5, 2, 1, 1, 4, 3, 4, 4, 1, 1], [ @';...
            ['"m:Steel", "", "' num2str(indrehudtykkelse) '", "", "", "", "", "", "", ""], "Surface'
num2str(Surface+1) ':' num2str(Surfacenew) '" )']};


        Curve = Curvenew;
        k = k+7;

        %Create mesh seed on the edges
        for i = Surface+1:Surfacenew
            File(k+1:k+4)={['mesh_seed_create( "Surface ' num2str(i) '.1", 2, 0, 0., '
num2str(Meshsizelangships) ', 0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.2", 2, 0, 0., ' num2str(Meshsizetverrships) ',
0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.4", 2, 0, 0., ' num2str(Meshsizetverrships) ',
0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.3", 2, 0, 0., ' num2str(Meshsizelangships) ',
0. )']};

            k = k+4;
        end

        %Mesh the inner hull plating
        File(k+1:k+5) = {'fem_create_mesh_surf_4( "IsoMesh", 49152,  @';...
            ['"Surface ' num2str(Surface+1) ':' num2str(Surfacenew) '" @'];...
            ', 1, ["0.2"], "Quad4", "#", "#", "Coord 0", "Coord 0",  @';...
            'fem_create_mesh_surfa_num_nodes, fem_create_mesh_surfa_num_elems,  @';...
            'fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )'};
        k = k+5;

        %Add boundary conditions
        File(k+1:k+4) = {'loadsbcs_create2( "BCinnerhull", "Displacement", "Nodal", "", "Static", [ @';...
            ['"Surface ' num2str(Surface+1) '.1 ' num2str(Surfacenew) '.3 ' num2str(Surface+1)  ':'
num2str(Surfacenew) '.2 ' num2str(Surface+1) ':' num2str(Surfacenew) '.4 '...
            '], "Geometry", "Coord 0", "1.", ["<0 0 0>",  @';...
            '"<0 0 0>", "<      >", "<      >"], ["", "", "", ""] )';...
            '$# Load/BC set "BC1" created.'};

        k = k+4;

        %Make new group for the web of the inner stiffeners
        Surface = Surfacenew;
        File(k+1:k+4) = {'ga_group_create( "Innerstiffenerweb" )';...
            'ga_group_current_set( "Innerstiffenerweb" )';...
            '$# Group "Innerstiffenerweb" created.';...
            'STRING asm_create_line_xyz_created_ids[VIRTUAL]'};

        k= k+4;


        %Draw lines for the web of the inner stiffeners and exclude the points were there are stringers
        for j = 0:antallstringer
            for i =
1+j*stringeravstand/stiveravstand:stringeravstand/stiveravstand+j*stringeravstand/stiveravstand-1
                File((i-j)*3-2+k:(i-j)*3+k) = {['asm_const_line_xyz( "' num2str(i+Curve-j) '", "<0 '
num2str(-stiverihoyde) ' 0>", "[' num2str(i*stiveravstand) ' ' num2str(avstandindreytre) ' 0]", "Coord 0",
@';...
                    'asm_create_line_xyz_created_ids )';...
                    ['$# 1 Line created: Line ' num2str(i+Curve-j)]};
            end
        end
```

Appendix: Matlab
Script for drawing and
running the analysis

```matlab
        Surfacenew = Surface + stringeravstand*(antallstringer+1)/stiveravstand - 1 - antallstringer;
        Curvenew = Curve + stringeravstand*(antallstringer+1)/stiveravstand - 1 - antallstringer;
        k = k+((antallstringer+1)*stringeravstand/stiveravstand-1-antallstringer)*3;

        %Extrude to curves to new surfaces
        File(1+k:k+7) = {'STRING sgm_sweep_surface_e_created_ids[VIRTUAL]';...
            ['sgm_const_surface_extrude( "' num2str(Surface+1) '", "<0 0 '
num2str(rammeavstand*(antallrammer+1)) '>", 1., 0., "[0 0 0]", "Coord 0",  @'];...
            ['"Curve ' num2str(Curve+1) ':' num2str(Curvenew) '", sgm_sweep_surface_e_created_ids )'];...
            ['$# ' num2str(Curvenew-Curve) ' Surfaces Created: Surfaces ' num2str(Surface+1) ':'
num2str(Surfacenew)];...
            'elementprops_create( "Innerstiffenerweb", 51, 25, 35, 11003, 1, 20, [13, 20, 36, 1004, @';...
            ' 11044, 11136, 11026, 11027, 1011, 11182], [5, 2, 1, 1, 4, 3, 4, 4, 1, 1], [ @';...
            ['"m:Steel", "", "' num2str(Indrestiverstegtykkelse) '", "", "", "", "", "", "", ""], "Surface'
num2str(Surface+1) ':' num2str(Surfacenew) '" )']};

        Curve = Curvenew;
        k = k+7;

        %Add mesh seed one the edges
        for i = Surface+1:Surfacenew
            File(k+1:k+4)={['mesh_seed_create( "Surface ' num2str(i) '.1", 2, 0, 0., '
num2str(Meshsizelangships) ', 0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.2", 2, 0, 0., ' num2str(Meshsizeinnerstif) ',
0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.4", 2, 0, 0., ' num2str(Meshsizeinnerstif) ',
0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.3", 2, 0, 0., ' num2str(Meshsizelangships) ',
0. )']};

            k = k+4;
        end

        %Mesh the web of the inner stiffeners
        File(k+1:k+5) = {'fem_create_mesh_surf_4( "IsoMesh", 49152,  @';...
            ['"Surface ' num2str(Surface+1) ':' num2str(Surfacenew) '" @'];...
            ', 1, ["0.2"], "Quad4", "#", "#", "Coord 0", "Coord 0",  @';...
            'fem_create_mesh_surfa_num_nodes, fem_create_mesh_surfa_num_elems,  @';...
            'fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )'};
        k = k+5;

        %Add boundary conditions
        File(k+1:k+4) = {'loadsbcs_create2( "BCinnerstiffenerweb", "Displacement", "Nodal", "", "Static",
[ @';...
            ['"Surface ' num2str(Surface+1)  ':' num2str(Surfacenew) '.2 ' num2str(Surface+1) ':'
num2str(Surfacenew) '.4 '...
            '], "Geometry", "Coord 0", "1.", ["<0 0 0>",  @';...
            '"<0 0 0>", "<      >", "<      >"], ["", "", "", ""] )';...
            '$# Load/BC set "BC1" created.'};

        k = k+4;
        Surface = Surfacenew;

        %Create a group for the flange of the inner stiffeners
        File(k+1:k+4) = {'ga_group_create( "Innerstiffenerflange" )';...
            'ga_group_current_set( "Innerstiffenerflange" )';...
            '$# Group "Innerstiffenerflange" created.';...
            'STRING asm_create_line_xyz_created_ids[VIRTUAL]'};

        k = k+4;

        %Draw lines for the flange of the inner stiffeners and exclude the points were there are stringers
        for j = 0:antallstringer
            for i =
1+j*stringeravstand/stiveravstand:stringeravstand/stiveravstand+j*stringeravstand/stiveravstand-1
                File((i-j)*3-2+k:(i-j)*3+k) = {['asm_const_line_xyz( "' num2str(i+Curve-j) '", "<'
num2str(stiveribredde) ' 0 0>", "[' num2str(i*stiveravstand) ' ' num2str(avstandindreytre-stiverihoyde) ' 0]",
"Coord 0",  @'];...
                    'asm_create_line_xyz_created_ids )';...
                    ['$# 1 Line created: Line ' num2str(i+Curve-j)]};
            end
        end
```

U
egian University of Science and Technology
artment of Marine Technology

Appendix: Matlab
Script for drawing and
running the analysis

```matlab
        Curvenew = Curve + stringeravstand*(antallstringer+1)/stiveravstand - 1 - antallstringer;
        Surfacenew = Surface + stringeravstand*(antallstringer+1)/stiveravstand - 1 - antallstringer;
        k = k+((antallstringer+1)*stringeravstand/stiveravstand-1-antallstringer)*3;

        %Extrude the lines to new surfaces
        File(1+k:k+6) = {['sgm_const_surface_extrude( "' num2str(Surface+1) '", "<0 0 '
num2str(rammeavstand*(antallrammer+1)) '>", 1., 0., "[0 0 0]", "Coord 0",  @'];...
            ['"Curve ' num2str(Curve+1) ':' num2str(Curvenew) '", sgm_sweep_surface_e_created_ids )'];...
            ['$# ' num2str(Curvenew-Curve) ' Surfaces Created: Surfaces ' num2str(Surface+1) ':'
num2str(Surfacenew)];...
            'elementprops_create( "Innerstiffenerflange", 51, 25, 35, 11003, 1, 20, [13, 20, 36, 1004, @';...
            ' 11044, 11136, 11026, 11027, 1011, 11182], [5, 2, 1, 1, 4, 3, 4, 4, 1, 1], [ @';...
            ['"m:Steel", "", "' num2str(Indrestiverflenstykkelse) '", "", "", "", "", "", ""], "Surface'
num2str(Surface+1) ':' num2str(Surfacenew) '" )']};


        Curve = Curvenew;
        k = k+6;

        %Add mesh seed to the edges
        for i = Surface+1:Surfacenew
            File(k+1:k+4)={['mesh_seed_create( "Surface ' num2str(i) '.1", 2, 0, 0., '
num2str(Meshsizelangships) ', 0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.2", 2, 0, 0., ' num2str(Meshsizetverrships) ',
0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.4", 2, 0, 0., ' num2str(Meshsizetverrships) ',
0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.3", 2, 0, 0., ' num2str(Meshsizelangships) ',
0. )']};

            k = k+4;
        end

        % Mesh the flange of the inner stiffeners
        File(k+1:k+5) = {'fem_create_mesh_surf_4( "IsoMesh", 49152,  @';...
            ['"Surface ' num2str(Surface+1) ':' num2str(Surfacenew) '" @'];...
            ', 1, ["0.2"], "Quad4", "#", "#", "Coord 0", "Coord 0",  @';...
            'fem_create_mesh_surfa_num_nodes, fem_create_mesh_surfa_num_elems,  @';...
            'fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )'};
        k = k+5;

        %Add boundary conditions
        File(k+1:k+4) = {'loadsbcs_create2( "BCinnerstiffeneflange", "Displacement", "Nodal", "", "Static",
[ @';...
            ['"Surface ' num2str(Surface+1)  ':' num2str(Surfacenew) '.2 ' num2str(Surface+1) ':'
num2str(Surfacenew) '.4" '...
            '], "Geometry", "Coord 0", "1.", ["<0 0 0>",  @'];...
            '"<0 0 0>", "<      >", "<      >"], ["", "", "", ""] )';...
            '$# Load/BC set "BC1" created.'};

        k = k+4;

        %Create a group  for the web frame stiffeners
        File(k+1:k+4) = {'ga_group_create( "Girderstiffener" )';...
            'ga_group_current_set( "Girderstiffener" )';...
            '$# Group "Girderstiffener" created.';...
            'STRING asm_create_line_xyz_created_ids[VIRTUAL]'};

        k = k+4;
        Surface = Surfacenew;

        %Draw lines for the web frame stiffeners on each web frame and excluding
        %the points were there are stringers
        for l = 1:antallrammer
            for j = 0:antallstringer
                for i = 1+j*stringeravstand/stiveravstand+(l-
1)*(stringeravstand/stiveravstand+antallstringer*stringeravstand/stiveravstand-
1):stringeravstand/stiveravstand+j*(stringeravstand/stiveravstand)+(l-
1)*(stringeravstand/stiveravstand+antallstringer*stringeravstand/stiveravstand-1)-1
                    x = (i-(l-1)*(stringeravstand/stiveravstand+antallstringer*stringeravstand/stiveravstand-
1))*stiveravstand;
                    z =l*rammeavstand;
```

Appendix: Matlab Script for drawing and running the analysis

```matlab
            File((i-j-antallstringer*(l-1))*3-2+k:(i-j-antallstringer*(l-1))*3+k) =
{['asm_const_line_xyz( "' num2str(i+Curve-j-antallstringer*(l-1)) '", "<0 0 ' num2str(rammestiverhoyde) '>",
"[' num2str(x) ' 0 ' num2str(z) ']", "Coord 0",  @'];...
                    'asm_create_line_xyz_created_ids )';...
                    ['$# 1 Line created: Line ' num2str(i+Curve-j-antallstringer*(l-1))]};
            end
        end
    end


    Surfacenew = Surface + (stringeravstand*(antallstringer+1)/stiveravstand -1-
antallstringer)*antallrammer;
    Curvenew = Curve + (stringeravstand*(antallstringer+1)/stiveravstand -1-antallstringer)*antallrammer;
    k = size(File,1);

    %Extruding the lines and add properties to the new surfaces
    File(1+k:k+6) = {['sgm_const_surface_extrude( "' num2str(Surface+1) '", "<0 '
num2str(avstandindreytre) ' 0>", 1., 0., "[0 0 0]", "Coord 0",  @'];...
        ['"Curve ' num2str(Curve+1) ':' num2str(Curvenew) '", sgm_sweep_surface_e_created_ids )'];...
        ['$# ' num2str(Curvenew-Curve) ' Surfaces Created: Surfaces ' num2str(Surface+1) ':'
num2str(Surfacenew)];...
        'elementprops_create( "Girderstiffener", 51, 25, 35, 11003, 1, 20, [13, 20, 36, 1004, @';...
        ' 11044, 11136, 11026, 11027, 1011, 11182], [5, 2, 1, 1, 4, 3, 4, 4, 1, 1], [ @';...
        ['"m:Steel", "", "' num2str(Girderstivertykkelse) '", "", "", "", "", "", "", ""], "Surface'
num2str(Surface+1) ':' num2str(Surfacenew) '" )']};

    k = k+6;

    %Add mesh seed to the edges
    for i = Surface+1:Surfacenew
        File(k+1:k+4)={['mesh_seed_create( "Surface ' num2str(i) '.2", 2, 0, 0., '
num2str(Meshsizelangships) ', 0. )'];...
            ['mesh_seed_create( "Surface ' num2str(i) '.1", 2, 0, 0., ' num2str(Meshsizetverrstringer) ',
0. )'];...
            ['mesh_seed_create( "Surface ' num2str(i) '.3", 2, 0, 0., ' num2str(Meshsizetverrstringer) ',
0. )'];...
            ['mesh_seed_create( "Surface ' num2str(i) '.4", 2, 0, 0., ' num2str(Meshsizelangships) ',
0. )']};
        k = k+4;
    end

    %Mesh the surfaces
    File(k+1:k+5) = {'fem_create_mesh_surf_4( "IsoMesh", 49152,  @';...
        ['"Surface ' num2str(Surface+1) ':' num2str(Surfacenew) '" @'];...
        ', 1, ["0.2"], "Quad4", "#", "#", "Coord 0", "Coord 0",  @';...
        'fem_create_mesh_surfa_num_nodes, fem_create_mesh_surfa_num_elems,  @';...
        'fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )'};
    k = k+5;

    Curve = Curvenew;
    Surface = Surfacenew;

    % Add lines for the flanges of the web frame stiffeners
    for l = 1:antallrammer
        for j = 0:antallstringer
            for i = 1+j*stringeravstand/stiveravstand+(l-
1)*(stringeravstand/stiveravstand+antallstringer*stringeravstand/stiveravstand-
1):stringeravstand/stiveravstand+j*(stringeravstand/stiveravstand)+(l-
1)*(stringeravstand/stiveravstand+antallstringer*stringeravstand/stiveravstand-1)-1
                x = (i-(l-1)*(stringeravstand/stiveravstand+antallstringer*stringeravstand/stiveravstand-
1))*stiveravstand;
                z =l*rammeavstand;
                File((i-j-antallstringer*(l-1))*3-2+k:(i-j-antallstringer*(l-1))*3+k) =
{['asm_const_line_xyz( "' num2str(i+Curve-j-antallstringer*(l-1)) '", "<' num2str(Girderstiverflenbredde) ' 0
0>", "[' num2str(x) ' 0 ' num2str(z+rammestiverhoyde) ']", "Coord 0",  @'];...
                    'asm_create_line_xyz_created_ids )';...
                    ['$# 1 Line created: Line ' num2str(i+Curve-j-antallstringer*(l-1))]};
            end
        end
    end


    Surfacenew = Surface + (stringeravstand*(antallstringer+1)/stiveravstand -1-
antallstringer)*antallrammer;
    Curvenew = Curve + (stringeravstand*(antallstringer+1)/stiveravstand -1-antallstringer)*antallrammer;
```

```matlab
        k = size(File,1);

        %Extrude the lines to new surfaces and give them properties
        File(1+k:k+6) = {['sgm_const_surface_extrude( "' num2str(Surface+1) '", "<0 '
num2str(avstandindreytre) ' 0>", 1., 0., "[0 0 0]", "Coord 0",  @'];...
            ['"Curve ' num2str(Curve+1) ':' num2str(Curvenew) '", sgm_sweep_surface_e_created_ids )'];...
            ['$# ' num2str(Curvenew-Curve) ' Surfaces Created: Surfaces ' num2str(Surface+1) ':'
num2str(Surfacenew)];...
            'elementprops_create( "Girderstiffenerflange", 51, 25, 35, 11003, 1, 20, [13, 20, 36, 1004, @';...
            ' 11044, 11136, 11026, 11027, 1011, 11182], [5, 2, 1, 1, 4, 3, 4, 4, 1, 1], [ @';...
            ['"m:Steel", "", "' num2str(Girderstiverflenstykkelse) '", "", "", "", "", "", "", ""], "Surface'
num2str(Surface+1) ':' num2str(Surfacenew) '" )']};

        k = k+6;

        % Add mesh seed to the edges
        for i = Surface+1:Surfacenew
            File(k+1:k+4)={['mesh_seed_create( "Surface ' num2str(i) '.2", 2, 0, 0., '
num2str(Meshsizetverrships) ', 0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.1", 2, 0, 0., ' num2str(Meshsizetverrstringer) ',
0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.3", 2, 0, 0., ' num2str(Meshsizetverrstringer) ',
0. )'];...
                ['mesh_seed_create( "Surface ' num2str(i) '.4", 2, 0, 0., ' num2str(Meshsizetverrships) ',
0. )']};
            k = k+4;
        end

        % Mesh the surfaces
        File(k+1:k+5) = {'fem_create_mesh_surf_4( "IsoMesh", 49152,  @';...
            ['"Surface ' num2str(Surface+1) ':' num2str(Surfacenew) '" @'];...
            ', 1, ["0.2"], "Quad4", "#", "#", "Coord 0", "Coord 0",  @';...
            'fem_create_mesh_surfa_num_nodes, fem_create_mesh_surfa_num_elems,  @';...
            'fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )'};
        k = k+5;

        %Create the LS-DYNA input file
        File(k+1:k+144) = {'ui_exec_function( "mesh_seed_display_mgr", "init" )';...
            'mesh_seed_display_mgr.erase(  )';...
            'REAL fem_equiv_all_x_equivtol_ab';...
            'INTEGER fem_equiv_all_x_segment';...
            'fem_equiv_all_group4( [" "], 0, "", 1, 1, 0.099999998, FALSE,  @';...
            'fem_equiv_all_x_equivtol_ab, fem_equiv_all_x_segment )';...
            'loadsbcs_eval_all(  )';...
            ['jobfile.open( "test", "ANALYZE NO JOBFILE" )'];...
            'jobfile.create_param( "version_opt", 0, 0., "LS-930", 4 )';...
            'jobfile.create_param( "sep_mesh", 0, 0., "OFF", 4 )';...
            'jobfile.create_param( "sep_mat", 0, 0., "OFF", 4 )';...
            'jobfile.create_param( "label_file", 0, 0., "*.sif", 4 )';...
            'jobfile.create_param( "list_file", 0, 0., "", 1 )';...
            'jobfile.create_param( "cpu_time_limit", 0, 0., "", 3 )';...
            'jobfile.create_param( "termination_step", 0, 0., "", 1 )';...
            'jobfile.create_param( "termination_time", 0, 0., "", 3 )';...
            'jobfile.create_param( "time_step_ratio_limit", 0, 0., "", 3 )';...
            'jobfile.create_param( "energy_ratio_limit", 0, 0., "", 3 )';...
            'jobfile.create_param( "mass_ratio_limit", 0, 0., "", 3 )';...
            'jobfile.create_param( "time_size_calc_basis", 0, 0., "Area/Edge", 4 )';...
            'jobfile.create_param( "initial_time_step", 0, 0., "", 3 )';...
            'jobfile.create_param( "step_scale_factor", 0, 0.89999998, "", 3 )';...
            'jobfile.create_param( "step_size_for_mass_scale", 0, 0., "", 3 )';...
            'jobfile.create_param( "shell_minimum_time_step", 0, 0., "", 3 )';...
            'jobfile.create_param( "load_curve_max_time_step", 0, 0., "", 1 )';...
            'jobfile.create_param( "erosion_flag", 0, 0., "OFF", 4 )';...
            'jobfile.create_param( "mass_scaling_first_step", 0, 0., "OFF", 4 )';...
            'jobfile.create_param( "Number_of_cpus", 1, 0., "", 1 )';...
            'jobfile.create_param( "One_rhs_only", 0, 0., "OFF", 4 )';...
            'jobfile.create_param( "Consistency_flg", 0, 0., "OFF", 4 )';...
            'jobfile.create_param( "parallel_force", 0, 0., "OFF", 4 )';...
            'jobfile.create_param( "optmenu_relaxation", 0, 0., "None.Active", 4 )';...
            'jobfile.create_param( "geometry_file", 0, 0., "", 4 )';...
            'jobfile.create_param( "relax_termination_time", 0, 1E+030, "", 3 )';...
            'jobfile.create_param( "con_tolerance", 0, 0.001, "", 3 )';...
            'jobfile.create_param( "Iterations_checks", 250, 0., "", 1 )';...
            'jobfile.create_param( "auto_control", 0, 0., "OFF", 4 )';...
```

NTNU
Norwegian University of Science and Technology
Department of Marine Technology

Appendix: Matlab
Script for drawing and
running the analysis

```
'jobfile.create_param( "papadrakakis", 0, 0.039999999, "", 3 )';...
'jobfile.create_param( "Relaxation_Factor", 0, 0.995, "", 3 )';...
'jobfile.create_param( "Time_scale_Factor", 0, 0.89999998, "", 3 )';...
'jobfile.create_param( "global_damping_curve", 0, 0., "f:ouatiafltlak", 4 )';...
'jobfile.create_param( "system_damping_constant", 0, 0., "", 3 )';...
'jobfile.create_param( "linear_viscosity_coefficient", 0, 0.059999999, "", 3 )';...
'jobfile.create_param( "quadratic_viscosity_coefficient", 0, 1.5, "", 3 )';...
'jobfile.create_param( "hourglass_viscosity_type", 0, 0., "LS_DYNA", 4 )';...
'jobfile.create_param( "hourglass_viscosity_coefficient", 0, 0.1, "", 3 )';...
'jobfile.create_param( "hourglass_energy_calc", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "stonewall_energy_diss", 0, 0., "ON", 4 )';...
'jobfile.create_param( "sliding_int_energy_diss", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "rayleigh_energy_diss", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Warning_Angle(iii)_warpage", 0, 20., "", 3 )';...
'jobfile.create_param( "Treat_degen_quads_as_tris", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "shell_theory", 0, 0., "Belytschko", 4 )';...
'jobfile.create_param( "warping_stiffness", 0, 0., "Belytschko-Tsay", 4 )';...
'jobfile.create_param( "normal_update", 0, 0., "Each Cycle", 4 )';...
'jobfile.create_param( "update_option", -1, 0., "", 1 )';...
'jobfile.create_param( "update_shell_thick", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "plastics_method", 0, 0., "Secant", 4 )';...
'jobfile.create_param( "consider_shell_thickness", 0, 0.,  @';...
'"Thickness not considered", 4 )';...
'jobfile.create_param( "check_penetration", 0, 0., "ON", 4 )';...
'jobfile.create_param( "max_check_multiplier", 0, 4., "", 3 )';...
'jobfile.create_param( "step_bet_search", 10, 0., "", 1 )';...
'jobfile.create_param( "search_old_surface", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "stiffness_value", 0, 0., "Min. of Master & Slave", 4 )';...
'jobfile.create_param( "scale_interface", 0, 0.1, "", 3 )';...
'jobfile.create_param( "scale_rigid_wall", 0, 0., "", 3 )';...
'jobfile.create_param( "shell_thick_include", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "auto_reorientation", 0, 0., "Active for Automated", 4 )';...
'jobfile.create_param( "control_subroutine", 0, 0., "", 1 )';...
'jobfile.create_param( "friction_subroutine", 0, 0., "", 1 )';...
'jobfile.create_param( "bin_state_time_int", 0, 0., "", 3 )';...
'jobfile.create_param( "excl_damp_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "local_coord_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "one_plot_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "elim_rigid_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "output_hglass_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "output_time_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "num_beam_int_dbox", 0, 0., "", 1 )';...
'jobfile.create_param( "inc_surf_strain_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "excl_sh_tensor_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "excl_sh_strain_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "excl_sh_res_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "excl_int_energy_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "num_var_sol_dbox", 0, 0., "", 1 )';...
'jobfile.create_param( "num_var_sh_dbox", 0, 0., "", 1 )';...
'jobfile.create_param( "num_sh_int_dbox", 0, 0., "", 1 )';...
'jobfile.create_param( "bin_history_time_int", 0, 0., "", 3 )';...
'jobfile.create_param( "inc_nodes_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_beams_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_shell_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_solid_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_th_sh_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_extra_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Pri_During_Input", 0, 0., "ON", 4 )';...
'jobfile.create_param( "Echo_File_Options", 0, 0., "Suppress Both", 4 )';...
'jobfile.create_param( "Velocities", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Update_Beam", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Interface_File", 0, 0., "", 3 )';...
'jobfile.create_param( "Interface_File_Name", 0, 0., "INTFOR", 4 )';...
'jobfile.create_param( "Time_Step", 0, 0., "ON", 4 )';...
'jobfile.create_param( "Time_Interval", 0, 0., "", 1 )';...
'jobfile.create_param( "Tolerance_Status", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Tolerance_Value", 0, 0., "", 3 )';...
'jobfile.create_param( "viewport", 0, 0., "default_viewport", 4 )';...
'jobfile.create_param( "datbox_groupname", 0, 0., "default_group", 4 )';...
'jobfile.create_param( "tranlational", 0, 0., "<0,0,0>", 4 )';...
'jobfile.create_param( "rotational", 0, 0., "<0,0,0>", 4 )';...
'jobfile.create_param( "loadcase_for_analysis", 0, 0., "Default", 4 )';...
'jobfile.create_param( "set_node_opt", 0, 0., "YES", 4 )';...
'jobfile.create_param( "set_beam_opt", 0, 0., "YES", 4 )';...
```

NU
egian University of Science and Technology
rtment of Marine Technology

Appendix: Matlab
Script for drawing and
running the analysis

```
    'jobfile.create_param( "set_discrete_opt", 0, 0., "YES", 4 )';...
    'jobfile.create_param( "set_shell_opt", 0, 0., "YES", 4 )';...
    'jobfile.create_param( "set_solid_opt", 0, 0., "YES", 4 )';...
    'jobfile.create_param( "set_tshell_opt", 0, 0., "YES", 4 )';...
    'jobfile.create_param( "w_db_node", 5, 0., "", 1 )';...
    'jobfile.create_param( "w_db_beam", 5, 0., "", 1 )';...
    'jobfile.create_param( "w_db_discrete", 5, 0., "", 1 )';...
    'jobfile.create_param( "w_db_shell", 5, 0., "", 1 )';...
    'jobfile.create_param( "w_db_solid", 5, 0., "", 1 )';...
    'jobfile.create_param( "w_db_tshell", 5, 0., "", 1 )';...
    'jobfile.create_param( "all_groups_in_db_id", 1, 0., "", 1 )';...
    'jobfile.create_param( "all_groups_in_db_id_1", 0, 0., "Girderstiffener", 101 )';...
    'jobfile.create_param( "setcard_fullmodel_group_id", 1, 0., "", 1 )';...
    'jobfile.create_param( "setcard_fullmodel_group_id_1", 0, 0., "Girderstiffener" @';...
    ', 101 )';...
    'jobfile.create_param( "factor_length_from_MADYMO", 0, 1., "", 3 )';...
    'jobfile.create_param( "factor_on_time_from_MADYMO", 0, 1., "", 3 )';...
    'jobfile.create_param( "factor_on_force_from_MADYMO", 0, 1., "", 3 )';...
    'jobfile.create_param( "Wait_time_as_MADYMO_computes", 0, 0., "", 3 )';...
    'jobfile.create_param( "Flip_X_coord_of_MADYMO", 0, 0., "OFF", 4 )';...
    'jobfile.create_param( "Flip_Y_coord_of_MADYMO", 0, 0., "OFF", 4 )';...
    'jobfile.create_param( "Flip_Z_coord_of_MADYMO", 0, 0., "OFF", 4 )';...
    'jobfile.create_param( "Num_Dyna_steps_per_MADYMO_step", 1, 0., "", 1 )';...
    'jobfile.create_param( "filename_interface", 0, 0.,  @';...
    '"Default_interfacefile.isf1", 4 )';...
    'jobfile.create_param( "restart_label_file", 0, 0., "", 4 )';...
    'jobfile.create_param( "DTIwrite_switch_id", 0, 0., "End Deck", 4 )';...
    'jobfile.create_param( "DTIdirect_text_toggle_id", 0, 0., "OFF", 4 )';...
    'jobfile.close(  )';...
    'uil_file_close.go(  )';...
    'sys_library( "add", "lsdyna3d.plb" )';...
    ['lsdyna3d_spawn_generic( "pat3lsdyna", " -d C:\Users\Torstein\Documents\utenis\' filnavn '" //
@'];...
    ['".db -j test", TRUE )']};

k = k+144;

%Create the file so it can be run in Patran
fid = fopen('Input.ses.01', 'wt');
fprintf(fid, '%s\n', File{:});
fclose(fid);

%Run Patran with the controll file
dos('C:\MSC.Software\Patran_x64\20121\bin\patran.exe -b -sfp
C:\Users\Torstein\Documents\utenis\input.ses.01');

%Make the controll file for the making of the iceberg
Fileis = {'$# Recorded by: Patran 2012 64-Bit';...
    'uil_file_new.go( "C:\MSC.Software\Patran_x64\20121/template.db",  @';...
    %Specify filename
    ['"C:\Users\Torstein\Documents\utenis\' filnavnIS '.db" )'];...
    '$# Question from application FILE';...
    %Overwrite if existing
    ['$#     Database C:\Users\Torstein\Documents\utenis\' filnavnIS '.db already exists.'];...
    '$# Do you wish to delete the existing database and create a new one ?';...
    '$? YES 36000002';...
    '$# Copying C:\MSC.Software\Patran_x64\20121/template.db to';...
    ['$# C:\Users\Torstein\Documents\utenis\' filnavnIS '.db'];...
    '$# Template copy complete.';...
    '$# Database version 3.12 created by Patran 2012 64-Bit successfully opened.';...
    ['$# Creating journal file C:\Users\Torstein\Documents\utenis\' filnavnIS '.db.jou at'];...
    '$# 28-Jan-13 17:37:19';...
    '$# Synchronizing the User Data Manager';...
    'set_current_dir( "C:\Users\Torstein\Documents\utenis" )';...
    '$# The current working directory has been changed to the database location';...
    '$# shown.';...
    '$# ';...
    '$#';...
    'uil_pref_analysis.set_analysis_preference( "LS-DYNA3D", "Structural", ".key",  @';...
    '"", "No Mapping" )';...
    '$#  Loading... ls_dyna3d_defn_dflt_tbs( LS-DYNA3D, Structural ));';...
    'ga_group_create( "Ice" )';...
    'ga_group_current_set( "Ice" )';...
    '$# Group "Ice" created.';...
```

```matlab
        'material.create( "Analysis code ID", 10001, "Analysis type ID", 1, "Ice", 1, @';...
        ' "", "Isotropic", 1, "Directionality", 1, "Linearity", 1, "Homogeneous", 0,  @';...
        '"Linear Elastic", 1, "Model Options & IDs", ["Linear Elastic(MAT1)", "Solid",  @';...
        '"", "", ""], [11002, 11003, 0, 0, 0], "Active Flag", 1, "Create", 11,  @';...
        '"External Flag", FALSE, "Property IDs", ["Density", "Elastic Modulus",  @';...
        '"Poisson Ratio"], [16, 2, 5, 0], "Property Values", ["900", "9E11", "0.3",  @';...
        '""] )';...
        'STRING asm_create_line_xyz_created_ids[VIRTUAL]';...
        'material.create( "Analysis code ID", 10001, "Analysis type ID", 1, "Pusher", 2, @';...
        ' "", "Isotropic", 1, "Directionality", 1, "Linearity", 1, "Homogeneous", 0,  @';...
        '"Linear Elastic", 1, "Model Options & IDs", ["Linear Elastic(MAT1)", "Solid",  @';...
        '"", "", ""], [11002, 11003, 0, 0, 0], "Active Flag", 1, "Create", 11,  @';...
        '"External Flag", FALSE, "Property IDs", ["Density", "Elastic Modulus",  @';...
        '"Poisson Ratio"], [16, 2, 5, 0], "Property Values", ["7890", "9E11", "0.3",  @';...
        '""] )'};

    k = 40;

    %Finding the radius of the lower end of the cone
    x1 = Radius(kkk) * cos(Angle(iii)*pi/180);

    %Finding the radius of the topp of the cone
    x2 = x1+H*tan(Angle(iii)*pi/180);

    %Finding the height the lower end of the cone must be for the radius to
    % reach exatly zero heigth
    y = Radius(kkk)*sin(Angle(iii)*pi/180);

    % finding the radius of inner sylinder used to eliminate the zero
    % lenght edge of the mesh in the centre of the iceberg
    Curve = 0;
    if Radius(kkk)*cos(asin((y*3+floor((Radius(kkk)-
y)/(Meshsize/3))*Meshsize)/(3*Radius(kkk))))>Radius(kkk)/500
        r=0.6*Radius(kkk)*cos(asin((y*3+floor((Radius(kkk)-y)/(Meshsize/3))*Meshsize)/(Radius(kkk)*3)));
    else
        r=0.6*Radius(kkk)*cos(asin((y*3+(floor((Radius(kkk)-y)/(Meshsize/3))-
3)*Meshsize)/(Radius(kkk)*3)));
    end

    %Making a line for the ice pusher plate
    Fileis(k+1:k+3) = {['asm_const_line_xyz( "' num2str(1) '", "<' num2str(x2-r) ' 0 0>", "[' num2str(r)
' ' num2str(H+Meshsize+Radius(kkk)-y) ' 0]", "Coord 0",  @'];...
        'asm_create_line_xyz_created_ids )';...
        ['$# 1 Line created: Line ' num2str(Curve+1)]};
    %making a line for the top of the cone
    Fileis(k+4:k+6) = {['asm_const_line_xyz( "' num2str(2) '", "<' num2str(x2-r) ' 0 0>", "[' num2str(r)
' ' num2str(H+Radius(kkk)-y) ' 0]", "Coord 0",  @'];...
        'asm_create_line_xyz_created_ids )';...
        ['$# 1 Line created: Line ' num2str(Curve+2)]};
    %Making a line for the lower end of the cone
    Fileis(k+7:k+9) = {['asm_const_line_xyz( "' num2str(3) '", "<' num2str(x1-r) ' 0 0>", "[' num2str(r)
' ' num2str(Radius(kkk)-y) ' 0]", "Coord 0",  @'];...
        'asm_create_line_xyz_created_ids )';...
        ['$# 1 Line created: Line ' num2str(Curve+3)]};


    Curve = 3;
    k = k+9;

    %Making lines with spacing equal to the mesh size to discibe the shape
    %of the radius
    for i=1:floor((Radius(kkk)-y)/Meshsize)
        %Braking if the lines become to short
        if Radius(kkk)*cos(asin((y+i*Meshsize)/Radius(kkk)))<Radius(kkk)/500
            break
        end
        Fileis(k+1:k+3) = {['asm_const_line_xyz( "' num2str(Curve+1) '", "<' num2str(-
r+Radius(kkk)*cos(asin((y+i*Meshsize)/Radius(kkk)))) ' 0 0>", "[' num2str(r) ' ' num2str(-
i*Meshsize+Radius(kkk)-y) ' 0]", "Coord 0",  @'];...
            'asm_create_line_xyz_created_ids )';...
            ['$# 1 Line created: Line ' num2str(Curve+1)]};
        k = k+3;
        Curve = Curve + 1;
    end
```

```matlab
%To get a sharper edge there is tried with lines with spacng equal to
%a tird of the element size in the front
for j=i*3+1:floor((Radius(kkk)-y)/(Meshsize/3))
    %Braking if the lines become to short
    if Radius(kkk)*cos(asin((y*3+j*Meshsize)/(3*Radius(kkk))))<Radius(kkk)/500
        break
    end
    Fileis(k+1:k+3) = {['asm_const_line_xyz( "' num2str(Curve+1) '", "<' num2str(-
r+Radius(kkk)*cos(asin((y*3+j*Meshsize)/(Radius(kkk)*3)))) ' 0 0>", "[' num2str(r) ' ' num2str(-
j*Meshsize/3+Radius(kkk)-y) ' 0]", "Coord 0",  @'];...
        'asm_create_line_xyz_created_ids )';...
        ['$# 1 Line created: Line ' num2str(Curve+1)]};
    k = k+3;
    Curve = Curve + 1;
end

Yttre = Curve;

%Make lines for internal surfaces
for i=1:floor((Radius(kkk)-y)/Meshsize)
    if Radius(kkk)*cos(asin((y+i*Meshsize)/Radius(kkk)))<Radius(kkk)/500
        break
    end
    Fileis(k+1:k+3) = {['asm_const_line_xyz( "' num2str(Curve+1) '", "<' num2str(r) ' 0 0>", "[0 '
num2str(-i*Meshsize+Radius(kkk)-y) ' 0]", "Coord 0",  @'];...
        'asm_create_line_xyz_created_ids )';...
        ['$# 1 Line created: Line ' num2str(Curve+1)]};
    k = k+3;
    Curve = Curve + 1;
    Hn = -i*Meshsize;
end
for j=i*3+1:floor((Radius(kkk)-y)/(Meshsize/3))
    if Radius(kkk)*cos(asin((y*3+j*Meshsize)/(3*Radius(kkk))))<Radius(kkk)/500
        break
    end
    Fileis(k+1:k+3) = {['asm_const_line_xyz( "' num2str(Curve+1) '", "<' num2str(r) ' 0 0>", "[0 '
num2str(-j*Meshsize/3+Radius(kkk)-y) ' 0]", "Coord 0",  @'];...
        'asm_create_line_xyz_created_ids )';...
        ['$# 1 Line created: Line ' num2str(Curve+1)]};
    k = k+3;
    Curve = Curve + 1;
    Hn = -j*Meshsize/3;
end

%Creating inner lines for the cone and the pusher
Fileis(k+1:k+3) = {['asm_const_line_xyz( "' num2str(Curve+1) '", "<' num2str(r) ' 0 0>", "[0 '
num2str(H+Meshsize+Radius(kkk)-y) ' 0]", "Coord 0",  @'];...
        'asm_create_line_xyz_created_ids )';...
        ['$# 1 Line created: Line ' num2str(Curve+1)]};
Fileis(k+4:k+6) = {['asm_const_line_xyz( "' num2str(Curve+2) '", "<' num2str(r) ' 0 0>", "[0 '
num2str(H+Radius(kkk)-y) ' 0]", "Coord 0",  @'];...
        'asm_create_line_xyz_created_ids )';...
        ['$# 1 Line created: Line ' num2str(Curve+2)]};
Fileis(k+7:k+9) = {['asm_const_line_xyz( "' num2str(Curve+3) '", "<' num2str(r) ' 0 0>", "[0 '
num2str(Radius(kkk)-y) ' 0]", "Coord 0",  @'];...
        'asm_create_line_xyz_created_ids )';...
        ['$# 1 Line created: Line ' num2str(Curve+3)]};

Curve = Curve +3;
k = k+9;

%Extruding all the lines in a circle
Fileis(k+1:k+3) = {'STRING sgm_sweep_surface_r_created_ids[VIRTUAL]';...
    ['sgm_const_surface_revolve( "1", "Coord 0.2", 360., 0., "Coord 0", "Curve 1:' num2str(Curve) '",
@'];...
    'sgm_sweep_surface_r_created_ids )'};

k = k+3;

Surface = Curve;
Solid = 0;
```

```matlab
%Creating solid from all external sufaces
for i=1:Yttre-1
    Fileis(k+1:k+3) = {'STRING sgm_solid_2surface_created_ids[VIRTUAL]';...
        ['sgm_const_solid_2surface_v1( "' num2str(Solid+1) '", TRUE, "Surface ' num2str(i) '",
"Surface ' num2str(i+1) '",  @'];...
            'sgm_solid_2surface_created_ids )'};
    Solid = Solid+1;
    k = k+3;
end
Indre = Solid+1;

%Creating solids of all inner surfaces
for i=Yttre+1:Surface-1
    Fileis(k+1:k+3) = {'STRING sgm_solid_2surface_created_ids[VIRTUAL]';...
        ['sgm_const_solid_2surface_v1( "' num2str(Solid+1) '", TRUE, "Surface ' num2str(i) '",
"Surface ' num2str(i+1) '",  @'];...
            'sgm_solid_2surface_created_ids )'};
    Solid = Solid+1;
    k = k+3;
end

%Add properties to the pusher plate
Fileis(k+1:k+2) = {'elementprops_create( "Pusher", 71, 25, 20, 11027, 1, 20, [13, 20, 1011, 11182],
[ @';...
        ['5, 2, 1, 1], ["Pusher", "", "", ""], "Solid 1 ' num2str(Indre) '" )']};

k = k+2;

%Add properties to the ice
Fileis(k+1:k+2) = {'elementprops_create( "Ice", 71, 25, 20, 11027, 1, 20, [13, 20, 1011, 11182],
[ @';...
        ['5, 2, 1, 1], ["Ice", "", "", ""], "Solid 2:' num2str(Indre-1) ' ' num2str(Indre+1) ':'
num2str(Solid) '" )']};

k = k+2;

%Mesh all external solids with iso mesh and sepperate the pusher plate
%in a group
Fileis(k+1:k+33) = {'INTEGER fem_create_mesh_solid_num_nodes';...
    'INTEGER fem_create_mesh_solid_num_elems';...
    'STRING fem_create_mesh_s_nodes_created[VIRTUAL]';...
    'STRING fem_create_mesh_s_elems_created[VIRTUAL]';...
    ['fem_create_mesh_sol_5( "Solid 2:' num2str(Indre-1) '", "IsoMesh", "Hex8", 1, ["'
num2str(Meshsize) '"], 49152, 0,  @'];...
    '1, 0, 1, 0., "", "#", "#", "Coord 0", "Coord 0",  @';...
    'fem_create_mesh_solid_num_nodes, fem_create_mesh_solid_num_elems,  @';...
    'fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )';...
    'fem_associate_elems_to_ep( "Ice", fem_create_mesh_s_elems_created,  @';...
    'fem_create_mesh_solid_num_elems )';...
    'mesh_seed_display_mgr.erase(  )';...
    'REAL fem_equiv_all_x_equivtol_ab';...
    'INTEGER fem_equiv_all_x_segment';...
    %Used eqivalence to connect nodes between solids
    'fem_equiv_all_group4( [" "], 0, "", 1, 1, 0.099999998, FALSE,  @';...
    'fem_equiv_all_x_equivtol_ab, fem_equiv_all_x_segment )';...
    'ga_group_create( "Pusher" )';...
    'ga_group_current_set( "Pusher" )';...
    '$# Group "Pusher" created.';...
    'INTEGER fem_create_mesh_solid_num_nodes';...
    'INTEGER fem_create_mesh_solid_num_elems';...
    'STRING fem_create_mesh_s_nodes_created[VIRTUAL]';...
    'STRING fem_create_mesh_s_elems_created[VIRTUAL]';...
    ['fem_create_mesh_sol_5( "Solid 1", "IsoMesh", "Hex8", 1, ["' num2str(Meshsize) '"], 49152, 0,
@'];...
    '1, 0, 1, 0., "", "#", "#", "Coord 0", "Coord 0",  @';...
    'fem_create_mesh_solid_num_nodes, fem_create_mesh_solid_num_elems,  @';...
    'fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )';...
    'fem_associate_elems_to_ep( "Pusher", fem_create_mesh_s_elems_created,  @';...
    'fem_create_mesh_solid_num_elems )';...
    'mesh_seed_display_mgr.erase(  )';...
    'REAL fem_equiv_group_x_equivtol_ab';...
    'INTEGER fem_equiv_group_x_segment';...
    'fem_equiv_all_group4( ["Pusher"], 0, "", 1, 1, 0.099999998, FALSE,  @';...
    'fem_equiv_all_x_equivtol_ab, fem_equiv_all_x_segment )'};
```

NU
egian University of Science and Technology
partment of Marine Technology

Appendix: Matlab
Script for drawing and
running the analysis

```matlab
        k = k+33;

        %Mesh all internal surfaces with paver and extrude into solid elements
        Fileis(k+1:k+26) = {'ga_group_current_set( "Ice" )';...
            'ui_exec_function( "mesh_seed_display_mgr", "init" )';...
            'INTEGER fem_create_mesh_surfa_num_nodes';...
            'INTEGER fem_create_mesh_surfa_num_elems';...
            'STRING fem_create_mesh_s_nodes_created[VIRTUAL]';...
            'STRING fem_create_mesh_s_elems_created[VIRTUAL]';...
            ['fem_create_mesh_surf_4( "Hybrid", 49680, "surface ' num2str(Surface-3) '", 4, ["'
num2str(Meshsize/2) '", "0.1",  @'];...
            '"0.2", "1.0"], "Quad4", "#", "#", "Coord 0", "Coord 0",  @';...
            'fem_create_mesh_surfa_num_nodes, fem_create_mesh_surfa_num_elems,  @';...
            'fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )';...
            '$? YES 2008020';...
            'INTEGER fem_sweep_elems_n_nodes_created';...
            'INTEGER fem_sweep_elems_n_elems_created';...
            'STRING fem_sweep_elems_ex_created_nids[VIRTUAL]';...
            'STRING fem_sweep_elems_ex_created_eids[VIRTUAL]';...
            ['fem_sweep_extrude_1( "#", "#", "Coord 0", "<0 1 0>", "' num2str(H-Hn) '", "0.0 ", 3,  @'];...
            ['"Surface ' num2str(Surface-3) '", 10, ["Bar2", "Quad4", "Quad8", "Quad12", "Wedge6", "Wedge15",
 @'];...
            '"Wedge24", "Hex8", "Hex20", "Hex32", "", "", "", "", "", "", "", "", "", "",  @';...
            '"", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "" @';...
            ', "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "",  @';...
            '"", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "" @';...
            ', "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "",  @';...
            '"", ""], "Coord 0", "Coord 0", "Uniform: Element Length", ["3", "1.5",  @';...
            ['"0.1", "0.2", "' num2str(Meshsize) '", "2", "", "", "", ""], fem_sweep_elems_n_nodes_created,
 @'];...
            'fem_sweep_elems_n_elems_created, fem_sweep_elems_ex_created_nids,  @';...
            'fem_sweep_elems_ex_created_eids )'};

        k = k+26;

        %Make LS-DYNA input file
        Fileis(k+1:k+144) = {'ui_exec_function( "mesh_seed_display_mgr", "init" )';...
            'mesh_seed_display_mgr.erase(  )';...
            'REAL fem_equiv_all_x_equivtol_ab';...
            'INTEGER fem_equiv_all_x_segment';...
            'fem_equiv_all_group4( [" "], 0, "", 1, 1, 0.099999998, FALSE,  @';...
            'fem_equiv_all_x_equivtol_ab, fem_equiv_all_x_segment )';...
            'loadsbcs_eval_all(  )';...
            ['jobfile.open( "testis", "ANALYZE NO JOBFILE" )'];...
            'jobfile.create_param( "version_opt", 0, 0., "LS-930", 4 )';...
            'jobfile.create_param( "sep_mesh", 0, 0., "OFF", 4 )';...
            'jobfile.create_param( "sep_mat", 0, 0., "OFF", 4 )';...
            'jobfile.create_param( "label_file", 0, 0., "*.sif", 4 )';...
            'jobfile.create_param( "list_file", 0, 0., "", 1 )';...
            'jobfile.create_param( "cpu_time_limit", 0, 0., "", 3 )';...
            'jobfile.create_param( "termination_step", 0, 0., "", 1 )';...
            'jobfile.create_param( "termination_time", 0, 0., "", 3 )';...
            'jobfile.create_param( "time_step_ratio_limit", 0, 0., "", 3 )';...
            'jobfile.create_param( "energy_ratio_limit", 0, 0., "", 3 )';...
            'jobfile.create_param( "mass_ratio_limit", 0, 0., "", 3 )';...
            'jobfile.create_param( "time_size_calc_basis", 0, 0., "Area/Edge", 4 )';...
            'jobfile.create_param( "initial_time_step", 0, 0., "", 3 )';...
            'jobfile.create_param( "step_scale_factor", 0, 0.89999998, "", 3 )';...
            'jobfile.create_param( "step_size_for_mass_scale", 0, 0., "", 3 )';...
            'jobfile.create_param( "shell_minimum_time_step", 0, 0., "", 3 )';...
            'jobfile.create_param( "load_curve_max_time_step", 0, 0., "", 1 )';...
            'jobfile.create_param( "erosion_flag", 0, 0., "OFF", 4 )';...
            'jobfile.create_param( "mass_scaling_first_step", 0, 0., "OFF", 4 )';...
            'jobfile.create_param( "Number_of_cpus", 1, 0., "", 1 )';...
            'jobfile.create_param( "One_rhs_only", 0, 0., "OFF", 4 )';...
            'jobfile.create_param( "Consistency_flg", 0, 0., "OFF", 4 )';...
            'jobfile.create_param( "parallel_force", 0, 0., "OFF", 4 )';...
            'jobfile.create_param( "optmenu_relaxation", 0, 0., "None.Active", 4 )';...
            'jobfile.create_param( "geometry_file", 0, 0., "", 4 )';...
            'jobfile.create_param( "relax_termination_time", 0, 1E+030, "", 3 )';...
            'jobfile.create_param( "con_tolerance", 0, 0.001, "", 3 )';...
            'jobfile.create_param( "Iterations_checks", 250, 0., "", 1 )';...
            'jobfile.create_param( "auto_control", 0, 0., "OFF", 4 )';...
```

Norwegian University of Science and Technology
Department of Marine Technology

Appendix: Matlab
Script for drawing and
running the analysis

```
'jobfile.create_param( "papadrakakis", 0, 0.039999999, "", 3 )';...
'jobfile.create_param( "Relaxation_Factor", 0, 0.995, "", 3 )';...
'jobfile.create_param( "Time_scale_Factor", 0, 0.89999998, "", 3 )';...
'jobfile.create_param( "global_damping_curve", 0, 0., "f:ouatiafltlak", 4 )';...
'jobfile.create_param( "system_damping_constant", 0, 0., "", 3 )';...
'jobfile.create_param( "linear_viscosity_coefficient", 0, 0.059999999, "", 3 )';...
'jobfile.create_param( "quadratic_viscosity_coefficient", 0, 1.5, "", 3 )';...
'jobfile.create_param( "hourglass_viscosity_type", 0, 0., "LS_DYNA", 4 )';...
'jobfile.create_param( "hourglass_viscosity_coefficient", 0, 0.1, "", 3 )';...
'jobfile.create_param( "hourglass_energy_calc", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "stonewall_energy_diss", 0, 0., "ON", 4 )';...
'jobfile.create_param( "sliding_int_energy_diss", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "rayleigh_energy_diss", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Warning_Angle(iii)_warpage", 0, 20., "", 3 )';...
'jobfile.create_param( "Treat_degen_quads_as_tris", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "shell_theory", 0, 0., "Belytschko", 4 )';...
'jobfile.create_param( "warping_stiffness", 0, 0., "Belytschko-Tsay", 4 )';...
'jobfile.create_param( "normal_update", 0, 0., "Each Cycle", 4 )';...
'jobfile.create_param( "update_option", -1, 0., "", 1 )';...
'jobfile.create_param( "update_shell_thick", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "plastics_method", 0, 0., "Secant", 4 )';...
'jobfile.create_param( "consider_shell_thickness", 0, 0.,  @';...
'"Thickness not considered", 4 )';...
'jobfile.create_param( "check_penetration", 0, 0., "ON", 4 )';...
'jobfile.create_param( "max_check_multiplier", 0, 4., "", 3 )';...
'jobfile.create_param( "step_bet_search", 10, 0., "", 1 )';...
'jobfile.create_param( "search_old_surface", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "stiffness_value", 0, 0., "Min. of Master & Slave", 4 )';...
'jobfile.create_param( "scale_interface", 0, 0.1, "", 3 )';...
'jobfile.create_param( "scale_rigid_wall", 0, 0., "", 3 )';...
'jobfile.create_param( "shell_thick_include", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "auto_reorientation", 0, 0., "Active for Automated", 4 )';...
'jobfile.create_param( "control_subroutine", 0, 0., "", 1 )';...
'jobfile.create_param( "friction_subroutine", 0, 0., "", 1 )';...
'jobfile.create_param( "bin_state_time_int", 0, 0., "", 3 )';...
'jobfile.create_param( "excl_damp_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "local_coord_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "one_plot_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "elim_rigid_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "output_hglass_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "output_time_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "num_beam_int_dbox", 0, 0., "", 1 )';...
'jobfile.create_param( "inc_surf_strain_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "excl_sh_tensor_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "excl_sh_strain_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "excl_sh_res_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "excl_int_energy_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "num_var_sol_dbox", 0, 0., "", 1 )';...
'jobfile.create_param( "num_var_sh_dbox", 0, 0., "", 1 )';...
'jobfile.create_param( "num_sh_int_dbox", 0, 0., "", 1 )';...
'jobfile.create_param( "bin_history_time_int", 0, 0., "", 3 )';...
'jobfile.create_param( "inc_nodes_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_beams_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_shell_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_solid_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_th_sh_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_extra_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Pri_During_Input", 0, 0., "ON", 4 )';...
'jobfile.create_param( "Echo_File_Options", 0, 0., "Suppress Both", 4 )';...
'jobfile.create_param( "Velocities", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Update_Beam", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Interface_File", 0, 0., "", 3 )';...
'jobfile.create_param( "Interface_File_Name", 0, 0., "INTFOR", 4 )';...
'jobfile.create_param( "Time_Step", 0, 0., "ON", 4 )';...
'jobfile.create_param( "Time_Interval", 0, 0., "", 1 )';...
'jobfile.create_param( "Tolerance_Status", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Tolerance_Value", 0, 0., "", 3 )';...
'jobfile.create_param( "viewport", 0, 0., "default_viewport", 4 )';...
'jobfile.create_param( "datbox_groupname", 0, 0., "default_group", 4 )';...
'jobfile.create_param( "tranlational", 0, 0., "<0,0,0>", 4 )';...
'jobfile.create_param( "rotational", 0, 0., "<0,0,0>", 4 )';...
'jobfile.create_param( "loadcase_for_analysis", 0, 0., "Default", 4 )';...
'jobfile.create_param( "set_node_opt", 0, 0., "YES", 4 )';...
'jobfile.create_param( "set_beam_opt", 0, 0., "YES", 4 )';...
```

U
egian University of Science and Technology
rtment of Marine Technology

Appendix: Matlab
Script for drawing and
running the analysis

```
        'jobfile.create_param( "set_discrete_opt", 0, 0., "YES", 4 )';...
        'jobfile.create_param( "set_shell_opt", 0, 0., "YES", 4 )';...
        'jobfile.create_param( "set_solid_opt", 0, 0., "YES", 4 )';...
        'jobfile.create_param( "set_tshell_opt", 0, 0., "YES", 4 )';...
        'jobfile.create_param( "w_db_node", 5, 0., "", 1 )';...
        'jobfile.create_param( "w_db_beam", 5, 0., "", 1 )';...
        'jobfile.create_param( "w_db_discrete", 5, 0., "", 1 )';...
        'jobfile.create_param( "w_db_shell", 5, 0., "", 1 )';...
        'jobfile.create_param( "w_db_solid", 5, 0., "", 1 )';...
        'jobfile.create_param( "w_db_tshell", 5, 0., "", 1 )';...
        'jobfile.create_param( "all_groups_in_db_id", 1, 0., "", 1 )';...
        'jobfile.create_param( "all_groups_in_db_id_1", 0, 0., "Girderstiffener", 101 )';...
        'jobfile.create_param( "setcard_fullmodel_group_id", 1, 0., "", 1 )';...
        'jobfile.create_param( "setcard_fullmodel_group_id_1", 0, 0., "Girderstiffener" @';...
        '", 101 )';...
        'jobfile.create_param( "factor_length_from_MADYMO", 0, 1., "", 3 )';...
        'jobfile.create_param( "factor_on_time_from_MADYMO", 0, 1., "", 3 )';...
        'jobfile.create_param( "factor_on_force_from_MADYMO", 0, 1., "", 3 )';...
        'jobfile.create_param( "Wait_time_as_MADYMO_computes", 0, 0., "", 3 )';...
        'jobfile.create_param( "Flip_X_coord_of_MADYMO", 0, 0., "OFF", 4 )';...
        'jobfile.create_param( "Flip_Y_coord_of_MADYMO", 0, 0., "OFF", 4 )';...
        'jobfile.create_param( "Flip_Z_coord_of_MADYMO", 0, 0., "OFF", 4 )';...
        'jobfile.create_param( "Num_Dyna_steps_per_MADYMO_step", 1, 0., "", 1 )';...
        'jobfile.create_param( "filename_interface", 0, 0.,  @';...
        '"Default_interfacefile.isf1", 4 )';...
        'jobfile.create_param( "restart_label_file", 0, 0., "", 4 )';...
        'jobfile.create_param( "DTIwrite_switch_id", 0, 0., "End Deck", 4 )';...
        'jobfile.create_param( "DTIdirect_text_toggle_id", 0, 0., "OFF", 4 )';...
        'jobfile.close(  )';...
        'uil_file_close.go(  )';...
        'sys_library( "add", "lsdyna3d.plb" )';...
        ['lsdyna3d_spawn_generic( "pat3lsdyna", " -d C:\Users\Torstein\Documents\utenis\' filnavnIS '" //
@'];...
        ['".db -j testis", TRUE )']};

    k = k+144;

    %Save controll file
    fid = fopen('Inputis.ses.01', 'wt');
    fprintf(fid, '%s\n', Fileis{:});
    fclose(fid);

    %Run controll file in patran
    dos('C:\MSC.Software\Patran_x64\20121\bin\patran.exe -b -sfp
C:\Users\Torstein\Documents\utenis\Inputis.ses.01');
    analysis = {};

    cd('C:\Users\Torstein\Documents\utenis\')

    %Open LS-DYNA iceberg inputfile
    fid = fopen('testis.key');
    w=0 ;

    %Put the inputfile into a matrix
    while 1
        w = w+1;
        B=fgets(fid);
        analysis(w)={B};
        %Break at the end of the file
        if B==-1
            break
        end
    end
    fclose(fid);
    Z = 1;
    %Scan for "PUSHER"
    GG = [80 117 115 104 101 114 13 10];
    for i=1:size(analysis,2)
        if size(analysis{i},2)==8
            Z = (double(analysis{i}) - GG);
            if Z(:) == 0
                zz = i;
                break
            end
        end
```

```matlab
        end
    end

    %Scan for "ice"
    GG = [36 13 10];
    for i=zz+1:size(analysis,2)
        if size(analysis{i},2)==3
            Z = (double(analysis{i}) - GG);
            if Z(:) == 0
                xx = i;
                break
            end
        end
    end

    %Reads the first and the last element numberof the pusher plate by use
    %of the line number given from the search above
    Element_number = strread(analysis{xx-1});
    Element_number_lastpusher = Element_number(1);
    Element_number = strread(analysis{zz+5});
    Element_number_firstpusher = Element_number(1);
    %Finds the last element of the ice
    Element_number = strread(analysis{zz-2});
    Element_number_lastice = Element_number(1);

    %Making a new controll file to give properties to the elements and mesh
    %the rest of the internal surfaces
    FileIS2 ={['uil_file_open.go( "C:\Users\Torstein\Documents\utenis\' filnavnIS '.db" )'];...
        'set_current_dir( "C:\Users\Torstein\Documents\uttenis" )';...
        'ga_group_current_set( "Ice" )';...
        'elementprops_create( "Ice", 71, 25, 20, 11027, 1, 20, [13, 20, 1011, 11182], [ @';...
        ['5, 2, 1, 1], ["m:Ice", "", "", ""], "Element 1:' num2str(Element_number_lastice) ' '
num2str(Element_number_lastpusher+1) ':" // @'];...
        '"#" )';...
        '$? YES 29002006';...
        'ga_group_current_set( "Pusher" )';...
        'ui_exec_function( "mesh_seed_display_mgr", "init" )';...
        'INTEGER fem_create_mesh_surfa_num_nodes';...
        'INTEGER fem_create_mesh_surfa_num_elems';...
        'STRING fem_create_mesh_s_nodes_created[VIRTUAL]';...
        'STRING fem_create_mesh_s_elems_created[VIRTUAL]';...
        ['fem_create_mesh_surf_4( "Hybrid", 49680, "surface ' num2str(Surface-1) '", 4, ["'
num2str(Meshsize/2) '", "0.1",  @'];...
        '"0.2", "1.0"], "Quad4", "#", "#", "Coord 0", "Coord 0",  @';...
        'fem_create_mesh_surfa_num_nodes, fem_create_mesh_surfa_num_elems,  @';...
        'fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )';...
        '$? YES 2008020';...
        'INTEGER fem_sweep_elems_n_nodes_created';...
        'INTEGER fem_sweep_elems_n_elems_created';...
        'STRING fem_sweep_elems_ex_created_nids[VIRTUAL]';...
        'STRING fem_sweep_elems_ex_created_eids[VIRTUAL]';...
        ['fem_sweep_extrude_1( "#", "#", "Coord 0", "<0 1 0>", "' num2str(Meshsize) '", "0.0 ", 3,
@'];...
        ['"Surface ' num2str(Surface-1) '", 10, ["Bar2", "Quad4", "Quad8", "Quad12", "Wedge6", "Wedge15",
@'];...
        '"Wedge24", "Hex8", "Hex20", "Hex32", "", "", "", "", "", "", "", "", "", "",  @';...
        '"", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "" @';...
        ', "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "",  @';...
        '"", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "" @';...
        ', "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "", "",  @';...
        '"", ""], "Coord 0", "Coord 0", "Uniform: Element Length", ["3", "1.5",  @';...
        ['"0.1", "0.2", "' num2str(Meshsize) '", "2", "", "", "", ""], fem_sweep_elems_n_nodes_created,
@'];...
        'fem_sweep_elems_n_elems_created, fem_sweep_elems_ex_created_nids,  @';...
        'fem_sweep_elems_ex_created_eids )';...
        'ui_exec_function( "mesh_seed_display_mgr", "init" )';...
        'mesh_seed_display_mgr.erase( )';...
        'REAL fem_equiv_all_x_equivtol_ab';...
        'INTEGER fem_equiv_all_x_segment';...
        'fem_equiv_all_group4( [" "], 0, "", 1, 1, 0.099999998, FALSE,  @';...
        'fem_equiv_all_x_equivtol_ab, fem_equiv_all_x_segment )';...
        'loadsbcs_eval_all( )';...
        ['jobfile.open( "testis", "ANALYZE NO JOBFILE" )'];...
        '$? YES 6016027';...
```

NTNU
Norwegian University of Science and Technology
Department of Marine Technology

Appendix: Matlab
Script for drawing and
running the analysis

```matlab
'jobfile.create_param( "version_opt", 0, 0., "LS-930", 4 )';...
'jobfile.create_param( "sep_mesh", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "sep_mat", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "label_file", 0, 0., "*.sif", 4 )';...
'jobfile.create_param( "list_file", 0, 0., "", 1 )';...
'jobfile.create_param( "cpu_time_limit", 0, 0., "", 3 )';...
'jobfile.create_param( "termination_step", 0, 0., "", 1 )';...
'jobfile.create_param( "termination_time", 0, 0., "", 3 )';...
'jobfile.create_param( "time_step_ratio_limit", 0, 0., "", 3 )';...
'jobfile.create_param( "energy_ratio_limit", 0, 0., "", 3 )';...
'jobfile.create_param( "mass_ratio_limit", 0, 0., "", 3 )';...
'jobfile.create_param( "time_size_calc_basis", 0, 0., "Area/Edge", 4 )';...
'jobfile.create_param( "initial_time_step", 0, 0., "", 3 )';...
'jobfile.create_param( "step_scale_factor", 0, 0.89999998, "", 3 )';...
'jobfile.create_param( "step_size_for_mass_scale", 0, 0., "", 3 )';...
'jobfile.create_param( "shell_minimum_time_step", 0, 0., "", 3 )';...
'jobfile.create_param( "load_curve_max_time_step", 0, 0., "", 1 )';...
'jobfile.create_param( "erosion_flag", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "mass_scaling_first_step", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Number_of_cpus", 1, 0., "", 1 )';...
'jobfile.create_param( "One_rhs_only", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Consistency_flg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "parallel_force", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "optmenu_relaxation", 0, 0., "None.Active", 4 )';...
'jobfile.create_param( "geometry_file", 0, 0., "", 4 )';...
'jobfile.create_param( "relax_termination_time", 0, 1E+030, "", 3 )';...
'jobfile.create_param( "con_tolerance", 0, 0.001, "", 3 )';...
'jobfile.create_param( "Iterations_checks", 250, 0., "", 1 )';...
'jobfile.create_param( "auto_control", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "papadrakakis", 0, 0.039999999, "", 3 )';...
'jobfile.create_param( "Relaxation_Factor", 0, 0.995, "", 3 )';...
'jobfile.create_param( "Time_scale_Factor", 0, 0.89999998, "", 3 )';...
'jobfile.create_param( "global_damping_curve", 0, 0., "f:ouatiafltlak", 4 )';...
'jobfile.create_param( "system_damping_constant", 0, 0., "", 3 )';...
'jobfile.create_param( "linear_viscosity_coefficient", 0, 0.059999999, "", 3 )';...
'jobfile.create_param( "quadratic_viscosity_coefficient", 0, 1.5, "", 3 )';...
'jobfile.create_param( "hourglass_viscosity_type", 0, 0., "LS_DYNA", 4 )';...
'jobfile.create_param( "hourglass_viscosity_coefficient", 0, 0.1, "", 3 )';...
'jobfile.create_param( "hourglass_energy_calc", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "stonewall_energy_diss", 0, 0., "ON", 4 )';...
'jobfile.create_param( "sliding_int_energy_diss", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "rayleigh_energy_diss", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Warning_Angle(iii)_warpage", 0, 20., "", 3 )';...
'jobfile.create_param( "Treat_degen_quads_as_tris", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "shell_theory", 0, 0., "Belytschko", 4 )';...
'jobfile.create_param( "warping_stiffness", 0, 0., "Belytschko-Tsay", 4 )';...
'jobfile.create_param( "normal_update", 0, 0., "Each Cycle", 4 )';...
'jobfile.create_param( "update_option", -1, 0., "", 1 )';...
'jobfile.create_param( "update_shell_thick", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "plastics_method", 0, 0., "Secant", 4 )';...
'jobfile.create_param( "consider_shell_thickness", 0, 0.,  @';...
'"Thickness not considered", 4 )';...
'jobfile.create_param( "check_penetration", 0, 0., "ON", 4 )';...
'jobfile.create_param( "max_check_multiplier", 0, 4., "", 3 )';...
'jobfile.create_param( "step_bet_search", 10, 0., "", 1 )';...
'jobfile.create_param( "search_old_surface", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "stiffness_value", 0, 0., "Min. of Master & Slave", 4 )';...
'jobfile.create_param( "scale_interface", 0, 0.1, "", 3 )';...
'jobfile.create_param( "scale_rigid_wall", 0, 0., "", 3 )';...
'jobfile.create_param( "shell_thick_include", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "auto_reorientation", 0, 0., "Active for Automated", 4 )';...
'jobfile.create_param( "control_subroutine", 0, 0., "", 1 )';...
'jobfile.create_param( "friction_subroutine", 0, 0., "", 1 )';...
'jobfile.create_param( "bin_state_time_int", 0, 0., "", 3 )';...
'jobfile.create_param( "excl_damp_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "local_coord_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "one_plot_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "elim_rigid_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "output_hglass_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "output_time_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "num_beam_int_dbox", 0, 0., "", 1 )';...
'jobfile.create_param( "inc_surf_strain_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "excl_sh_tensor_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "excl_sh_strain_tg", 0, 0., "OFF", 4 )';...
```

NTNU
Norwegian University of Science and Technology
Department of Marine Technology

Appendix: Matlab
Script for drawing and
running the analysis

```
'jobfile.create_param( "excl_sh_res_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "excl_int_energy_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "num_var_sol_dbox", 0, 0., "", 1 )';...
'jobfile.create_param( "num_var_sh_dbox", 0, 0., "", 1 )';...
'jobfile.create_param( "num_sh_int_dbox", 0, 0., "", 1 )';...
'jobfile.create_param( "bin_history_time_int", 0, 0., "", 3 )';...
'jobfile.create_param( "inc_nodes_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_beams_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_shell_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_solid_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_th_sh_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_extra_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Pri_During_Input", 0, 0., "ON", 4 )';...
'jobfile.create_param( "Echo_File_Options", 0, 0., "Suppress Both", 4 )';...
'jobfile.create_param( "Velocities", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Update_Beam", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Interface_File", 0, 0., "", 3 )';...
'jobfile.create_param( "Interface_File_Name", 0, 0., "INTFOR", 4 )';...
'jobfile.create_param( "Time_Step", 0, 0., "ON", 4 )';...
'jobfile.create_param( "Time_Interval", 0, 0., "", 1 )';...
'jobfile.create_param( "Tolerance_Status", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Tolerance_Value", 0, 0., "", 3 )';...
'jobfile.create_param( "viewport", 0, 0., "default_viewport", 4 )';...
'jobfile.create_param( "datbox_groupname", 0, 0., "default_group", 4 )';...
'jobfile.create_param( "tranlational", 0, 0., "<0,0,0>", 4 )';...
'jobfile.create_param( "rotational", 0, 0., "<0,0,0>", 4 )';...
'jobfile.create_param( "loadcase_for_analysis", 0, 0., "Default", 4 )';...
'jobfile.create_param( "set_node_opt", 0, 0., "YES", 4 )';...
'jobfile.create_param( "set_beam_opt", 0, 0., "YES", 4 )';...
'jobfile.create_param( "set_discrete_opt", 0, 0., "YES", 4 )';...
'jobfile.create_param( "set_shell_opt", 0, 0., "YES", 4 )';...
'jobfile.create_param( "set_solid_opt", 0, 0., "YES", 4 )';...
'jobfile.create_param( "set_tshell_opt", 0, 0., "YES", 4 )';...
'jobfile.create_param( "w_db_node", 5, 0., "", 1 )';...
'jobfile.create_param( "w_db_beam", 5, 0., "", 1 )';...
'jobfile.create_param( "w_db_discrete", 5, 0., "", 1 )';...
'jobfile.create_param( "w_db_shell", 5, 0., "", 1 )';...
'jobfile.create_param( "w_db_solid", 5, 0., "", 1 )';...
'jobfile.create_param( "w_db_tshell", 5, 0., "", 1 )';...
'jobfile.create_param( "all_groups_in_db_id", 1, 0., "", 1 )';...
'jobfile.create_param( "all_groups_in_db_id_1", 0, 0., "Girderstiffener", 101 )';...
'jobfile.create_param( "setcard_fullmodel_group_id", 1, 0., "", 1 )';...
'jobfile.create_param( "setcard_fullmodel_group_id_1", 0, 0., "Girderstiffener" @';...
', 101 )';...
'jobfile.create_param( "factor_length_from_MADYMO", 0, 1., "", 3 )';...
'jobfile.create_param( "factor_on_time_from_MADYMO", 0, 1., "", 3 )';...
'jobfile.create_param( "factor_on_force_from_MADYMO", 0, 1., "", 3 )';...
'jobfile.create_param( "Wait_time_as_MADYMO_computes", 0, 0., "", 3 )';...
'jobfile.create_param( "Flip_X_coord_of_MADYMO", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Flip_Y_coord_of_MADYMO", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Flip_Z_coord_of_MADYMO", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Num_Dyna_steps_per_MADYMO_step", 1, 0., "", 1 )';...
'jobfile.create_param( "filename_interface", 0, 0.,  @';...
'"Default_interfacefile.isf1", 4 )';...
'jobfile.create_param( "restart_label_file", 0, 0., "", 4 )';...
'jobfile.create_param( "DTIwrite_switch_id", 0, 0., "End Deck", 4 )';...
'jobfile.create_param( "DTIdirect_text_toggle_id", 0, 0., "OFF", 4 )';...
'jobfile.close(  )';...
'uil_file_close.go(  )';...
'sys_library( "add", "lsdyna3d.plb" )';...
['lsdyna3d_spawn_generic( "pat3lsdyna", " -d C:\Users\Torstein\Documents\utenis\' filnavnIS '" //
@'];...
['".db -j testis", TRUE )']};

%Save controll file
fid = fopen('Inputis.ses.01', 'wt');
fprintf(fid, '%s\n', FileIS2{:});
fclose(fid);

%Run controll file in patran
dos('C:\MSC.Software\Patran_x64\20121\bin\patran.exe -b -sfp
C:\Users\Torstein\Documents\utenis\Inputis.ses.01');

%open the input file
```

86

U
egian University of Science and Technology
rtment of Marine Technology

Appendix: Matlab
Script for drawing and
running the analysis

```matlab
analysis = {};
fid = fopen('testis.key');
w=0 ;
%Add to a matrix
while 1
    w = w+1;
    B=fgets(fid);
    analysis(w)={B};
    if B==-1
        break
    end
end
fclose(fid);
Z = 0;
GG = [80 117 115 104 101 114 13 10];
for i=1:size(analysis,2)
    if size(analysis{i},2)==8
        Z = (double(analysis{i}) - GG);
        if Z(:)==0
            zz = i;
            break
        end
    end
end

Z=0;
GG = [36 13 10];
for i=zz+1:size(analysis,2)
    if size(analysis{i},2)==3
        Z = (double(analysis{i}) - GG);
        if Z(:)==0
            xx = i;
            break
        end
    end
end

Element_number = strread(analysis{zz-2});
Element_number_lastice = Element_number(1);

%Add properties to the last elements of the pusher plate
FileIS2 ={['uil_file_open.go( "C:\Users\Torstein\Documents\' filnavnIS '.db" )'];...
    'set_current_dir( "C:\Users\Torstein\Documents\utenis" )';...
    'ga_group_current_set( "Pusher" )';...
    'elementprops_create( "Pusher", 71, 25, 20, 11027, 1, 20, [13, 20, 1011, 11182], [ @';...
    ['5, 2, 1, 1], ["m:Pusher", "", "", ""], "Element ' num2str(Element_number_firstpusher) ':'
num2str(Element_number_lastpusher) ' ' num2str(Element_number_lastice+1) ':" // @'];...
    '"#" )';...
    '$? YES 29002006 ';...
    'jobfile.open( "testis", "ANALYZE NO JOBFILE" )';...
    '$? YES 6016027';...
    'jobfile.create_param( "version_opt", 0, 0., "LS-930", 4 )';...
    'jobfile.create_param( "sep_mesh", 0, 0., "OFF", 4 )';...
    'jobfile.create_param( "sep_mat", 0, 0., "OFF", 4 )';...
    'jobfile.create_param( "label_file", 0, 0., "*.sif", 4 )';...
    'jobfile.create_param( "list_file", 0, 0., "", 1 )';...
    'jobfile.create_param( "cpu_time_limit", 0, 0., "", 3 )';...
    'jobfile.create_param( "termination_step", 0, 0., "", 1 )';...
    'jobfile.create_param( "termination_time", 0, 0., "", 3 )';...
    'jobfile.create_param( "time_step_ratio_limit", 0, 0., "", 3 )';...
    'jobfile.create_param( "energy_ratio_limit", 0, 0., "", 3 )';...
    'jobfile.create_param( "mass_ratio_limit", 0, 0., "", 3 )';...
    'jobfile.create_param( "time_size_calc_basis", 0, 0., "Area/Edge", 4 )';...
    'jobfile.create_param( "initial_time_step", 0, 0., "", 3 )';...
    'jobfile.create_param( "step_scale_factor", 0, 0.89999998, "", 3 )';...
    'jobfile.create_param( "step_size_for_mass_scale", 0, 0., "", 3 )';...
    'jobfile.create_param( "shell_minimum_time_step", 0, 0., "", 3 )';...
    'jobfile.create_param( "load_curve_max_time_step", 0, 0., "", 1 )';...
    'jobfile.create_param( "erosion_flag", 0, 0., "OFF", 4 )';...
    'jobfile.create_param( "mass_scaling_first_step", 0, 0., "OFF", 4 )';...
    'jobfile.create_param( "Number_of_cpus", 1, 0., "", 1 )';...
    'jobfile.create_param( "One_rhs_only", 0, 0., "OFF", 4 )';...
    'jobfile.create_param( "Consistency_flg", 0, 0., "OFF", 4 )';...
    'jobfile.create_param( "parallel_force", 0, 0., "OFF", 4 )';...
```

NTNU
Norwegian University of Science and Technology
Department of Marine Technology

Appendix: Matlab
Script for drawing and
running the analysis

```
'jobfile.create_param( "optmenu_relaxation", 0, 0., "None.Active", 4 )';...
'jobfile.create_param( "geometry_file", 0, 0., "", 4 )';...
'jobfile.create_param( "relax_termination_time", 0, 1E+030, "", 3 )';...
'jobfile.create_param( "con_tolerance", 0, 0.001, "", 3 )';...
'jobfile.create_param( "Iterations_checks", 250, 0., "", 1 )';...
'jobfile.create_param( "auto_control", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "papadrakakis", 0, 0.039999999, "", 3 )';...
'jobfile.create_param( "Relaxation_Factor", 0, 0.995, "", 3 )';...
'jobfile.create_param( "Time_scale_Factor", 0, 0.89999998, "", 3 )';...
'jobfile.create_param( "global_damping_curve", 0, 0., "f:ouatiafltlak", 4 )';...
'jobfile.create_param( "system_damping_constant", 0, 0., "", 3 )';...
'jobfile.create_param( "linear_viscosity_coefficient", 0, 0.059999999, "", 3 )';...
'jobfile.create_param( "quadratic_viscosity_coefficient", 0, 1.5, "", 3 )';...
'jobfile.create_param( "hourglass_viscosity_type", 0, 0., "LS_DYNA", 4 )';...
'jobfile.create_param( "hourglass_viscosity_coefficient", 0, 0.1, "", 3 )';...
'jobfile.create_param( "hourglass_energy_calc", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "stonewall_energy_diss", 0, 0., "ON", 4 )';...
'jobfile.create_param( "sliding_int_energy_diss", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "rayleigh_energy_diss", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Warning_Angle(iii)_warpage", 0, 20., "", 3 )';...
'jobfile.create_param( "Treat_degen_quads_as_tris", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "shell_theory", 0, 0., "Belytschko", 4 )';...
'jobfile.create_param( "warping_stiffness", 0, 0., "Belytschko-Tsay", 4 )';...
'jobfile.create_param( "normal_update", 0, 0., "Each Cycle", 4 )';...
'jobfile.create_param( "update_option", -1, 0., "", 1 )';...
'jobfile.create_param( "update_shell_thick", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "plastics_method", 0, 0., "Secant", 4 )';...
'jobfile.create_param( "consider_shell_thickness", 0, 0.,  @';...
'"Thickness not considered", 4 )';...
'jobfile.create_param( "check_penetration", 0, 0., "ON", 4 )';...
'jobfile.create_param( "max_check_multiplier", 0, 4., "", 3 )';...
'jobfile.create_param( "step_bet_search", 10, 0., "", 1 )';...
'jobfile.create_param( "search_old_surface", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "stiffness_value", 0, 0., "Min. of Master & Slave", 4 )';...
'jobfile.create_param( "scale_interface", 0, 0.1, "", 3 )';...
'jobfile.create_param( "scale_rigid_wall", 0, 0., "", 3 )';...
'jobfile.create_param( "shell_thick_include", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "auto_reorientation", 0, 0., "Active for Automated", 4 )';...
'jobfile.create_param( "control_subroutine", 0, 0., "", 1 )';...
'jobfile.create_param( "friction_subroutine", 0, 0., "", 1 )';...
'jobfile.create_param( "bin_state_time_int", 0, 0., "", 3 )';...
'jobfile.create_param( "excl_damp_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "local_coord_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "one_plot_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "elim_rigid_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "output_hglass_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "output_time_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "num_beam_int_dbox", 0, 0., "", 1 )';...
'jobfile.create_param( "inc_surf_strain_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "excl_sh_tensor_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "excl_sh_strain_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "excl_sh_res_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "excl_int_energy_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "num_var_sol_dbox", 0, 0., "", 1 )';...
'jobfile.create_param( "num_var_sh_dbox", 0, 0., "", 1 )';...
'jobfile.create_param( "num_sh_int_dbox", 0, 0., "", 1 )';...
'jobfile.create_param( "bin_history_time_int", 0, 0., "", 3 )';...
'jobfile.create_param( "inc_nodes_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_beams_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_shell_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_solid_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_th_sh_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "inc_extra_tg", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Pri_During_Input", 0, 0., "ON", 4 )';...
'jobfile.create_param( "Echo_File_Options", 0, 0., "Suppress Both", 4 )';...
'jobfile.create_param( "Velocities", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Update_Beam", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Interface_File", 0, 0., "", 3 )';...
'jobfile.create_param( "Interface_File_Name", 0, 0., "INTFOR", 4 )';...
'jobfile.create_param( "Time_Step", 0, 0., "ON", 4 )';...
'jobfile.create_param( "Time_Interval", 0, 0., "", 1 )';...
'jobfile.create_param( "Tolerance_Status", 0, 0., "OFF", 4 )';...
'jobfile.create_param( "Tolerance_Value", 0, 0., "", 3 )';...
'jobfile.create_param( "viewport", 0, 0., "default_viewport", 4 )';...
```

U
egian University of Science and Technology
partment of Marine Technology

Appendix: Matlab
Script for drawing and
running the analysis

```matlab
        'jobfile.create_param( "datbox_groupname", 0, 0., "default_group", 4 )';...
        'jobfile.create_param( "tranlational", 0, 0., "<0,0,0>", 4 )';...
        'jobfile.create_param( "rotational", 0, 0., "<0,0,0>", 4 )';...
        'jobfile.create_param( "loadcase_for_analysis", 0, 0., "Default", 4 )';...
        'jobfile.create_param( "set_node_opt", 0, 0., "YES", 4 )';...
        'jobfile.create_param( "set_beam_opt", 0, 0., "YES", 4 )';...
        'jobfile.create_param( "set_discrete_opt", 0, 0., "YES", 4 )';...
        'jobfile.create_param( "set_shell_opt", 0, 0., "YES", 4 )';...
        'jobfile.create_param( "set_solid_opt", 0, 0., "YES", 4 )';...
        'jobfile.create_param( "set_tshell_opt", 0, 0., "YES", 4 )';...
        'jobfile.create_param( "w_db_node", 5, 0., "", 1 )';...
        'jobfile.create_param( "w_db_beam", 5, 0., "", 1 )';...
        'jobfile.create_param( "w_db_discrete", 5, 0., "", 1 )';...
        'jobfile.create_param( "w_db_shell", 5, 0., "", 1 )';...
        'jobfile.create_param( "w_db_solid", 5, 0., "", 1 )';...
        'jobfile.create_param( "w_db_tshell", 5, 0., "", 1 )';...
        'jobfile.create_param( "all_groups_in_db_id", 1, 0., "", 1 )';...
        'jobfile.create_param( "all_groups_in_db_id_1", 0, 0., "Girderstiffener", 101 )';...
        'jobfile.create_param( "setcard_fullmodel_group_id", 1, 0., "", 1 )';...
        'jobfile.create_param( "setcard_fullmodel_group_id_1", 0, 0., "Girderstiffener" @';...
        ', 101 )';...
        'jobfile.create_param( "factor_length_from_MADYMO", 0, 1., "", 3 )';...
        'jobfile.create_param( "factor_on_time_from_MADYMO", 0, 1., "", 3 )';...
        'jobfile.create_param( "factor_on_force_from_MADYMO", 0, 1., "", 3 )';...
        'jobfile.create_param( "Wait_time_as_MADYMO_computes", 0, 0., "", 3 )';...
        'jobfile.create_param( "Flip_X_coord_of_MADYMO", 0, 0., "OFF", 4 )';...
        'jobfile.create_param( "Flip_Y_coord_of_MADYMO", 0, 0., "OFF", 4 )';...
        'jobfile.create_param( "Flip_Z_coord_of_MADYMO", 0, 0., "OFF", 4 )';...
        'jobfile.create_param( "Num_Dyna_steps_per_MADYMO_step", 1, 0., "", 1 )';...
        'jobfile.create_param( "filename_interface", 0, 0.,  @';...
        '"Default_interfacefile.isf1", 4 )';...
        'jobfile.create_param( "restart_label_file", 0, 0., "", 4 )';...
        'jobfile.create_param( "DTIwrite_switch_id", 0, 0., "End Deck", 4 )';...
        'jobfile.create_param( "DTIdirect_text_toggle_id", 0, 0., "OFF", 4 )';...
        'jobfile.close(  )';...
        'uil_file_close.go(  )';...
        'sys_library( "add", "lsdyna3d.plb" )';...
        ['lsdyna3d_spawn_generic( "pat3lsdyna", " -d C:\Users\Torstein\Documents\utenis\' filnavnIS '" //
@'];...
        ['".db -j testis", TRUE )']};

    %Save controll file
    fid = fopen('Inputis.ses.01', 'wt');
    fprintf(fid, '%s\n', FileIS2{:});
    fclose(fid);

    %Run the controll file
    dos('C:\MSC.Software\Patran_x64\20121\bin\patran.exe -b -sfp
C:\Users\Torstein\Documents\utenis\Inputis.ses.01');

    %Run a controllfile in prepost to define all variables in there
    dos('C:\LS-PrePost\3.2-X64\lsprepost c=C:\Users\Torstein\Documents\utenis\lspostIS.cfile -nographics')
    cd('C:\Users\Torstein\Documents\');

    %Add a new begining to the file so it fit with the older version of
    %LS-DYNA that is on the server
    fid=fopen('begynnelse.k');
    v = 0;
    while 1
        v = v+1;
        B = fgets(fid);
        analysis(v) = {B};
        if B == -1
            break
        end
    end

    fclose(fid);
    cd('C:\Users\Torstein\Documents\utenis\');
    fid=fopen('testIS.k');
    %Remove the old begining
    for i=1:28
        B=fgets(fid);
    end
```

U
...egian University of Science and Technology
...rtment of Marine Technology

Appendix: Matlab
Script for drawing and
running the analysis

```matlab
%Add the rest of the file
while 1
    v = v+1;
    B=fgets(fid);
    analysis(v)={B};
    if B==-1
        break
    end
end
z=0;

%Search for some strange sign prepost add to the material model and
%remove them so prepost and LS-DYNA can read the material model
GG = [42 77 65 84 95 85 83 69 82 95 68 69 70 73 78 69 68 95 77 65 84 69 82 73 65 76 95 77 79 68 69 76
83 95 84 73 84 76 69 10];
for i=1:size(analysis,2)
    if size(analysis{i},2)==40
        Z = (double(analysis{i}) - GG);
        if Z(:)==0
            z = z+1;
            zz(z) =i;
        end
    end
end

analysis{zz(1)+11}=[];
analysis{zz(1)+10}=[];
analysis{zz(1)+9}=['1         0.01' char(10)];
analysis{zz(2)+11}=[];
analysis{zz(2)+10}=[];
analysis{zz(2)+9}=['0.7       1' char(10)];
%close the file
fclose(fid);

%Save the file with correct name
[fid, message] = fopen([filnavn '.k'], 'wt');
for i=1:size(analysis,2)
    fprintf(fid, '%s', analysis{i});
end

fclose(fid);

%Chack and remove earlier copy of the file
winscp = ['c:\WinSCP\winscp.com - batch /console /command "open *******:*********@129.241.140.53" "cd
analyser" "rm ""/home/torstemy/analyser/utenis2/' filnavn '.k""" "exit"'];
dos(winscp)

%Transfer the input file
winscp = ['c:\WinSCP\winscp.com - batch /console /command "open *******:*********@129.241.140.53" "cd
analyser" "put ""C:\Users\Torstein\Documents\utenis\' filnavn '.k"" ""/home/torstemy/analyser/utenis2/'
filnavn '.k""" "exit"'];
dos(winscp)

%Run the input file in LS-DYNA on the server
plink = ['c:\plink.exe  -l ******* -pw ******** ********@129.241.140.53 cd analyser ; cd utenis2;
export LSTC_LICENSE=network ; export LSTC_LICENSE_SERVER=129.241.140.40 ; /home/ls971
i=/home/torstemy/analyser/utenis2/' num2str(filnavn) '.k NCPU=2'];

dos(plink)


% Transfer the result to a network drive
cd('Y:\torstein\analyser\utenis\')

winscp = ['c:\WinSCP\winscp.com /console /command "open *******:*********@129.241.140.53" "get
/home/torstemy/analyser/utenis2 Y:\Torstein\analyser\utenis\' num2str(Angle(iii)) 'stiveravstand\" "exit"'];

dos(winscp)

    end
end
```

B. Appendix: Matlab script for result prossesing

```matlab
timestep = 0.002;
endtime = 0.6;
speed = 10;

%Choose the directories that shall be read
dir={'0.877rett' ; '15rammeavstand4' ; '20rammeavstand4'; '30rammeavstand4'; '45rammeavstand4';
'50stiveravstand'; '60stiveravstand'};
inpact_time(1:size(dir))=1;
%Read ncforc
for z=1:5


    %open directory
    cd(['Y:\Torstein\Analyser\1C\' dir{z} '\1C' ])
    % Open the output file
    fid = fopen('ncforc');

    %Removes the top lines of the file
    for i=1:15
        B = fgets(fid);
        Dummy = sscanf(B, '%s');
    end

    t = 0;
    if any(strcmp(who,'Areal'));
    else
        Areal(1,1:size(dir))=0;
    end

    while 1
        t = t+1;
        if B==-1
            break
        end
        %Removes lines between timesteps
        if t>1
            for s=1:6
                B = fgetl(fid);
                Dummy = sscanf(B, '%s');
            end
        end

        i=0;
        F=[];
        Coord=[];
        %Read values for each node in one timestep
        while 1
            i=i+1;
            B = fgetl(fid);

            % Ends if all nodes are read
            if isempty(B)
                break
            end
            if B==-1
                break
            end

            A = sscanf(B,'%e');
            B = fgetl(fid);
            C = sscanf(B,'%e');

            %Collect all data in the timestep
            F(i,:) = A;
            coord(i,:) = C;
            Coord(i,1) = coord(i,1);
            Coord(i,2) = coord(i,3);
            Coord(i,3) = coord(i,2);
            clearvars('coord')
```

```matlab
            % Finds elements with contact pressure and add to the area
            if F(i,5)~=0
                if size(Areal(:,z),1)<t
                    Areal(t,z)=0;
                end
                Areal(t,z)=Areal(t,z)+sqrt(F(i,2)^2+F(i,3)^2+F(i,4)^2)/F(i,5);
            end
        end


        % Resulting force
        SF(t,:,z)=sum(F);
        % Maximum force and pressure
        if sum(F)==0
        else
            SFM(t,:,z)=max(abs(F));
        end

        %Find the time step where the impact happens
        Coordx=0;
        Coordx = sortrows(Coord);
        if size(Coordx(:,:),1)>0
            if SF(t,3,z)==0 && (inpact_time(z)==t || inpact_time(z)==t-1)
                inpact_time(z) = t+1;
                y(t,z) = 0;

                %If the impact happend at this timestep add the value of the
                %highest coordinate in y-direction of the iceberg to the variable
            else if t==inpact_time(z)
                    y(t,z) = max(Coordx(:,3));

                    %After impact take maksimum coordinate in y-direction and subtract
                    %the coordinate at impact
                else
                    y(t,z) = max(Coordx(:,3))-y(inpact_time(z),z);
                end
            end
        end

        %Make plot over contact pressures at the different coordinates
        if t>=inpact_time(z)
            %If there is no force give NaN values to all points, so they become
            %empty in the result plot
            if size(F)==[0 0]
                Pressureplot(1:size(Pressureplot,1),1:size(Pressureplot,2),t,z)=NaN;
            else
                %find the nodes without pressure
                remove = find(F(:,5)==0);
                %Remove nodes witout pressure
                Coord(remove,:)=[];
                F(remove,:)=[];

                %Make an empty plot in the right size
                [xq,yq] = meshgrid(9:.05:14, 9:.05:14);

                %If there is no nodes set all values to nan
                if size(Coord(:,1),1)==0 || size(Coord(:,2),1)==0 || size(Coord(:,1),1)<4 ||
size(Coord(:,2),1)<4
                    Pressureplot(1:size(Pressureplot,1),1:size(Pressureplot,2),t,z)=NaN;
                    clearvars('F')
                else
                    %Set values to the values in the empty plot based on
                    %interpolating the values in the nodes
                    if size(F,1)>size(Coord,1)
                        Pressureplot(:,:,t,z) =
griddata(Coord(1:size(F,1),1),Coord(1:size(F,1),2),F(:,5),xq,yq);
                        clearvars('F')
                    else
                        Pressureplot(:,:,t,z) = griddata(Coord(:,1),Coord(:,2),F(1:size(Coord,1),5),xq,yq);
                        clearvars('F')
                    end
                end
            end
        end
    end
end
```

```
fclose(fid);

%Set the initial deformation to zero
if inpact_time(z)>0
    y(1:inpact_time(z),z)=0;
end

%Only allow rising deformaton on the shipside between timesteps
%Because the deformation er read of the nodes of the iceberg and will
%give wrong value for each layer of eroded elements
for i=2:size(y,1)
    if y(i,z)<y(i-1,z)
        y(i,z)=y(i-1,z);
    end
end

%Calculating the deformation on the iceberg by use of the speed and
%deformation on the shipside
y_iceberg(1:size(y),z) = timestep*speed*(1:size(y,1));
for q=1:size(y,1)
    yiceberg(q,z) = y_iceberg(q,z)-y_iceberg(inpact_time(z),z)-y(q,z);
end

%Set initial deforation of the iceberg to zero
yiceberg(1:inpact_time(z),z)=0;

% Sort the area matrix in increasing order and collect the indexes
[d(z,1:size(Areal,1)),ind(1:size(Areal,1))]=sort(abs(Areal(:,z)),1);
for i=1:size(ind,2)
    % Calculate average pressure and sort it according to area matrix
    avpres(z,i)=abs(SF(ind(i),3,z)/Areal(ind(i),z));
end

% Assuming normal distrubution of pressure verses area and find the
% mean value and standard deviation
for i=0:-1+size(ind,2)/5
    if i==-1+size(ind,2)/5
        [m(i+1),s(i+1)]=normfit(avpres(z,1+5*i:4+5*i));
        A(i+1)=mean(d(z,1+5*i:4+5*i));
    else
        [m(i+1),s(i+1)]=normfit(avpres(z,1+5*i:5+5*i));
        A(i+1)=mean(d(z,1+5*i:5+5*i));
    end
end

% Fit curves to mean and to mean pluss 3 times the standard deviation
% using least squares method
C(1:2,z)=polyfit(log(A)',log(m+3*s),1)';
V(1:2,z)=polyfit(log(d(z,:)),log(avpres(z,:)),1)';

% making the graphs values
x=0.001:0.001:5;
Press(:,1,z)=exp(C(2,1))*x.^C(1,z);
Press(:,2,z)=exp(V(2,z))*x.^V(1,z);
ISO=7.4*10^6*x.^-0.7;

%Read matsum
fid = fopen('matsum');

%Removes the top lines of the file
for i=1:21
    B = fgets(fid);
    Dummy = sscanf(B, '%s');
end

t=0;

while 1
    t = t+1;

    %Removes lines between timesteps
    if t>1
        for s=1:35
            B = fgetl(fid);
            if B==-1
```

```matlab
                break
            end
            Dummy = sscanf(B, '%s');
        end
    end

    B = fgetl(fid);
    if B==-1
        break
    end

    %Energy(t,:) = sscanf(B,'%*s %e %*s %e %*s %e %*s %e %*s %e');

    end
    fclose(fid)

    %Totenergy = Energy(:,2)+Energy(:,4);
    % L = 0;
    %
    % for i=1:size(Totenergy)-1
    %     if SF(i,2,z)==0
    %         DeltaL(i)=0;
    %     else
    %         DeltaL(i) = (Totenergy(i+1)-Totenergy(i))/SF(i,2,z);
    %     end
    %     L(i+1) = L(i)+DeltaL(i);
    % end
end

%Plot the pressure area graph
figure(1)
hold on
xlabel('Area [m^2]')
ylabel('Average pressure [Pa]')
scatter(d(1,:),avpres(1,:),'.b')
scatter(d(2,:),avpres(2,:),'.g')
scatter(d(3,:),avpres(3,:),'.r')
scatter(d(4,:),avpres(4,:),'.c')
plot(x,ISO,'-r','LineWidth',2)
axis([0,3,0,7*10^7])
legend({dir{1:4} 'ISO'})
hold off

%Plot the Displacment-mpact force graph
Legend = {};
dir=strrep(dir,'rett','');
for i=1:size(y,2)
    Legend{i} = [dir{i} ' m'];
end
colors=['b', 'g', 'r', 'c', 'k'];
figure(1)
hold on
xlabel('Displacment [m]')
ylabel('Impact force [N]')
for ss=1:size(y,2)
    plot(y(2:(find(yiceberg(:,ss)>1,1)),ss),-SF(2:(find(yiceberg(:,ss)>1,1)),3,ss), colors(ss))
end
for ss=1:size(y,2)
    plot(-yiceberg(2:(find(yiceberg(:,ss)>1,1)),ss),-SF(2:(find(yiceberg(:,ss)>1,1)),3,ss), colors(ss))
end
axis([-1 0.3 0 1.1*max(max(-SF(2:100,3,:)))])
set(gca,'XTick',[-1:0.1:0 0.1:0.1:0.3])
hleg=legend(Legend);
set(hleg,'Location','NorthEast')
hold off

%Plot pressure plot
figure(3)
Title={'10^{\circ}' '15^{\circ}' '20^{\circ}' '30^{\circ}' '40^{\circ}' '50^{\circ}' '60^{\circ}'};
for sss=1:5
    subplot(3,3,sss)
    surface(xq,yq,Pressureplot(:,:,find(yiceberg(:,sss)>0.054,1),sss))
    xlabel('x-coordinate')
    ylabel('y-coordinate')
    ss=0;
    while 1
```

```matlab
        line([12+0.8*ss 12+0.8*ss], [min(yq(:,1)) max(yq(:,1))], [100 100], 'linestyle', '-')
        line([11.2-0.8*ss 11.2-0.8*ss], [min(yq(:,1)) max(yq(:,1))], [100 100], 'linestyle', '-')
        if max(xq(1,:))<12+0.8*(ss+1)
            break
        end
        ss=ss+1;
    end
    %line([min(xq(1,:)) max(xq(1,:))], [12.8 12.8], [100 100])
    %line([min(xq(1,:)) max(xq(1,:))], [9.6 9.6], [100 100])
    axis([min(xq(1,:))+2 max(xq(1,:))-1.7 min(yq(:,1))+2 max(yq(:,1))-1.7])
    caxis([0 1*10^7])
    title(Title{sss})
end


%Creating the energy variables
ENERGYICE(1,1:size(y,2))=0;
ENERGYSHIP(1,1:size(y,2))=0;

%Calculating energy as the area below the impact force displacment graph
for i=1:size(y,1)-1
    for j=1:size(y,2)
        if yiceberg(i,j)<1.5
            ENERGYSHIP(i+1,j) = ENERGYSHIP(i,j)+(y(1+i,j)-y(i,j))*SF(1+i,3,j);
            ENERGYICE(i+1,j) = ENERGYICE(i,j)+(yiceberg(1+i,j)-yiceberg(i,j))*SF(1+i,3,j);
        else
            ENERGYSHIP(i+1,j) = ENERGYSHIP(i,j);
            ENERGYICE(i+1,j) = ENERGYICE(i,j);
        end
    end
end
hold on

%Plot the energy of the ship against energy of the ceberg
plot(-ENERGYICE,-ENERGYSHIP)
%legend(Title)
ylabel('Energy absorbed by the ship [J]')
xlabel('Energy absorbed by the iceberg [J]')
axis([0 7*10^5 0 11*10^4])
hold off

%Sumarizing to find total energy
toten=ENERGYSHIP+ENERGYICE;
%Ploting total energy against deformation of the shipside
figure(12)
hold on
for q=1:size(y,2)

    plot(-toten(1:find(ENERGYICE(:,q)<(-1*10^6)),q),y(1:find(ENERGYICE(:,q)<(-1*10^6)),q), colors(q))
    mellom=strrep(strrep(dir{q},'rett',''), 's', '');
    Title{q} = [mellom(1:min(5,size(mellom,2))) ' m'];
end
xlabel('Total Energy [J]')
ylabel('Deformation ship [m]')
% axis([0 2*10^6 0 0.2])
Title={'10^{\circ}' '15^{\circ}' '20^{\circ}' '30^{\circ}' '45^{\circ}' '50^{\circ}' '60^{\circ}'};
legend(Title)
```

C. Appendix: Controll file for LS-prepost

```
$# LS-PrePost command file created by LS-PrePost 3.2 -27Oct2012(15:20) -64bit-Window
$# Created on Mar-08-2013 (15:04:26)
openc d3plot "-classic"
open keyword "C:\Users\Torstein\Documents\1C\testis.key"
save keywordoutversion 4
import keyword offset
shad
import keyword "C:\Users\Torstein\Documents\1C\test.key"
save keywordoutversion 4
genselect target node
occfilter clear
occfilter clear
genselect node add part 1/0
genselect node add part 2/0
rotate_model 0 0 0 x 180
translate_model 11.65 -0.01 11.7
genselect target node
occfilter clear
genselect clear
genselect clear
genselect clear
setpart loaddata
genselect part add part 8/0
genselect target part
occfilter clear
genselect target part
occfilter clear
genselect clear
genselect clear
genselect part add part 1/0
setpart createset 1 1 0 0 0 0 "Ice"
genselect clear
genselect part add part 2/0
setpart createset 2 1 0 0 0 0 "Pusher"
genselect clear
genselect part add part 3/0
genselect part add part 4/0
genselect part add part 5/0
genselect part add part 6/0
genselect part add part 7/0
genselect part add part 8/0
genselect part add part 9/0
genselect part add part 10/0
genselect part add part 11/0
setpart createset 3 1 0 0 0 0 "Ship"
genselect clear
KEYWORD INPUT 1
*CONTACT_AUTOMATIC_SINGLE_SURFACE_ID
$#     cid                                    title
      1Shipinternal
$#   ssid     msid   sstyp   mstyp  sboxid  mboxid     spr     mpr
        3        0       2       0       0       0       0       0
```

96

NTNU
Norwegian University of Science and Technology
Department of Marine Technology

```
$#     fs      fd      dc      vc     vdc   penchk     bt      dt
 0.300000   0.000   0.000   0.000   0.000       0   0.0001.0000E+20
$#    sfs     sfm     sst     mst    sfst    sfmt     fsf     vsf
 1.000000 1.000000   0.000   0.000 1.000000 1.000000 1.000000 1.000000
*SET_PART_LIST_TITLE
Ship
$#    sid     da1     da2     da3     da4  solver
     3   0.000   0.000   0.000   0.000MECH
$#   pid1    pid2    pid3    pid4    pid5    pid6    pid7    pid8
     3       0       0       0       0       0       0       0
*END
keyword updatekind
CONTACT_AUTOMATIC_SINGLE_SURFACE
KEYWORD INPUT 2
*CONTACT_ERODING_SINGLE_SURFACE_ID
$#   cid                                      title
     2Ice
$#   ssid    msid    sstyp   mstyp  sboxid  mboxid     spr     mpr
     1       0       2       0       0       0       0       0
$#     fs      fd      dc      vc     vdc   penchk     bt      dt
 0.150000   0.000   0.000   0.000   0.000       0   0.0001.0000E+20
$#    sfs     sfm     sst     mst    sfst    sfmt     fsf     vsf
 1.000000 1.000000   0.000   0.000 1.000000 1.000000 1.000000 1.000000
$#   isym   erosop    iadj
     0       1       0
*SET_PART_LIST_TITLE
Ice
$#    sid     da1     da2     da3     da4  solver
     1   0.000   0.000   0.000   0.000MECH
$#   pid1    pid2    pid3    pid4    pid5    pid6    pid7    pid8
     1       0       0       0       0       0       0       0
*END
keyword updatekind
CONTACT_ERODING_SINGLE_SURFACE
KEYWORD INPUT 3
*CONTACT_ERODING_SURFACE_TO_SURFACE_ID
$#   cid                                      title
     3Collision
$#   ssid    msid    sstyp   mstyp  sboxid  mboxid     spr     mpr
     1       3       2       2       0       0       1       0
$#     fs      fd      dc      vc     vdc   penchk     bt      dt
 0.150000   0.000   0.000   0.000   0.000       0   0.0001.0000E+20
$#    sfs     sfm     sst     mst    sfst    sfmt     fsf     vsf
 1.000000 1.000000   0.000   0.000 1.000000 1.000000 1.000000 1.000000
$#   isym   erosop    iadj
     0       1       1
*SET_PART_LIST_TITLE
Ice
$#    sid     da1     da2     da3     da4  solver
     1   0.000   0.000   0.000   0.000MECH
$#   pid1    pid2    pid3    pid4    pid5    pid6    pid7    pid8
     1       0       0       0       0       0       0       0
*SET_PART_LIST_TITLE
Ship
$#    sid     da1     da2     da3     da4  solver
     3   0.000   0.000   0.000   0.000MECH
```

egian University of Science and Technology

rtment of Marine Technology

```
$#   pid1   pid2   pid3   pid4   pid5   pid6   pid7   pid8
      3      4      5      6      7      8      9     10
$#   pid1   pid2   pid3   pid4   pid5   pid6   pid7   pid8
     11      0      0      0      0             0      0      0
*END
keyword updatekind
CONTACT_ERODING_SURFACE_TO_SURFACE
KEYWORD INPUT 4
*CONTACT_ERODING_SURFACE_TO_SURFACE_ID
$#   cid                                     title
      4Pusher
$#   ssid   msid   sstyp   mstyp  sboxid  mboxid    spr     mpr
      1      2      2      2      0      0      0      0
$#    fs     fd     dc     vc    vdc  penchk     bt     dt
    0.150  0.000  0.000  0.000  0.000      0   0.0001.0000E+20
$#   sfs    sfm    sst    mst    sfst   sfmt    fsf    vsf
  1.000000 1.000000   0.000    0.000 1.000000 1.000000 1.000000 1.000000
$#   isym  erosop   iadj
      0      1      1
*SET_PART_LIST_TITLE
Ice
$#   sid    da1    da2    da3    da4  solver
      1    0.000  0.000  0.000  0.000MECH
$#   pid1   pid2   pid3   pid4   pid5   pid6   pid7   pid8
      1      0      0      0      0      0      0      0
*SET_PART_LIST_TITLE
Pusher
$#   sid    da1    da2    da3    da4  solver
      2    0.000  0.000  0.000  0.000MECH
$#   pid1   pid2   pid3   pid4   pid5   pid6   pid7   pid8
      2      0      0      0      0      0      0      0
*END
keyword updatekind
CONTACT_ERODING_SURFACE_TO_SURFACE
KEYWORD INPUT 3
*SECTION_SHELL
$#  secid  elform    shrf    nip  propt qr/irid   icomp   setyp
      3      2   0.000      5      1      0      0      1
$#   t1     t2     t3     t4    nloc   marea    idof  edgset
  0.020000 0.020000 0.020000 0.020000   0.000   0.000   0.000      0
*END
keyword updatekind
SECTION_SHELL
KEYWORD INPUT 4
*SECTION_SHELL
$#  secid  elform    shrf    nip  propt qr/irid   icomp   setyp
      4      2   0.000      5      1      0      0      1
$#   t1     t2     t3     t4    nloc   marea    idof  edgset
  0.012000 0.012000 0.012000 0.012000   0.000   0.000   0.000      0
*END
keyword updatekind
SECTION_SHELL
KEYWORD INPUT 5
*SECTION_SHELL
$#  secid  elform    shrf    nip  propt qr/irid   icomp   setyp
      5      2   0.000      5      1      0      0      1
```

```
$#    t1      t2      t3      t4     nloc    marea    idof   edgset
 0.018000 0.018000 0.018000 0.018000   0.000    0.000    0.000      0
*END
keyword updatekind
SECTION_SHELL
KEYWORD INPUT 6
*SECTION_SHELL
$#  secid   elform    shrf     nip   propt   qr/irid   icomp   setyp
      6      2    0.000     5      1      0       0       1
$#    t1      t2      t3      t4     nloc    marea    idof   edgset
 0.015000 0.015000 0.015000 0.015000   0.000    0.000    0.000      0
*END
keyword updatekind
SECTION_SHELL
KEYWORD INPUT 7
*SECTION_SHELL
$#  secid   elform    shrf     nip   propt   qr/irid   icomp   setyp
      7      2    0.000     5      1      0       0       1
$#    t1      t2      t3      t4     nloc    marea    idof   edgset
 0.012000 0.012000 0.012000 0.012000   0.000    0.000    0.000      0
*END
keyword updatekind
SECTION_SHELL
KEYWORD INPUT 8
*SECTION_SHELL
$#  secid   elform    shrf     nip   propt   qr/irid   icomp   setyp
      8      2    0.000     5      1      0       0       1
$#    t1      t2      t3      t4     nloc    marea    idof   edgset
 0.021000 0.021000 0.021000 0.021000   0.000    0.000    0.000      0
*END
keyword updatekind
SECTION_SHELL
KEYWORD INPUT 9
*SECTION_SHELL
$#  secid   elform    shrf     nip   propt   qr/irid   icomp   setyp
      9      2    0.000     5      1      0       0       1
$#    t1      t2      t3      t4     nloc    marea    idof   edgset
 0.012000 0.012000 0.012000 0.012000   0.000    0.000    0.000      0
*END
keyword updatekind
SECTION_SHELL
KEYWORD INPUT 10
*SECTION_SHELL
$#  secid   elform    shrf     nip   propt   qr/irid   icomp   setyp
     10      2    0.000     5      1      0       0       1
$#    t1      t2      t3      t4     nloc    marea    idof   edgset
 0.015000 0.015000 0.015000 0.015000   0.000    0.000    0.000      0
*END
keyword updatekind
SECTION_SHELL
KEYWORD INPUT 11
*SECTION_SHELL
$#  secid   elform    shrf     nip   propt   qr/irid   icomp   setyp
     11      2    0.000     5      1      0       0       1
$#    t1      t2      t3      t4     nloc    marea    idof   edgset
 0.012000 0.012000 0.012000 0.012000   0.000    0.000    0.000      0
```

NTNU
Norwegian University of Science and Technology
Department of Marine Technology

```
*END
keyword updatekind
SECTION_SHELL
KEYWORD INPUT 1
$ This is the unit load curve
*DEFINE_CURVE_TITLE
Speed
$#   lcid    sidr    sfa    sfo    offa    offo   dattyp
     1      0   0.000   0.000   0.000   0.000      0
$#        a1           o1
      0.000         0.000
    0.001000      10.000000
    2.000000      10.000000
*END
keyword updatekind
DEFINE_CURVE
KEYWORD INPUT 1
*BOUNDARY_PRESCRIBED_MOTION_RIGID_ID
$#   id                              heading
    1Speed
$#   pid    dof    vad    lcid    sf    vid   death   birth
     2      2      0      1  1.000000     01.0000E+28   0.000
*END
keyword updatekind
BOUNDARY_PRESCRIBED_MOTION_RIGID
Keyword Delete 288 1 2 3
KEYWORD INPUT 1
$ Material : Ice
*MAT_USER_DEFINED_MATERIAL_MODELS_TITLE
Ice
$#   mid     ro     mt     lmc    nhv   iortho   ibulk    ig
     1 900.0000     41      16     20      0       3      4
$#  ivect  ifail  itherm  ihyper   ieos    lmca   unused   unused
      0      1      0       0       0       8       0       0
$# p1      p2     p3     p4     p5     p6     p7     p8
9.5E9    0.3   7.91667E10 3.6538E9 22.92900E6  2.06E6  -0.0229    -2E6
$# p1      p2     p3     p4     p5     p6     p7     p8
1     0.01
*END
keyword updatekind
MAT_USER_DEFINED_MATERIAL_MODELS
KEYWORD INPUT 2
$ Material : Pusher
*MAT_RIGID_TITLE
pusher
$#   mid     ro     e     pr     n    couple     m    alias
     2 900.00000 9.0000E+9  0.300000   0.000    0.000   0.000
$#   cmo    con1    con2
      1       6       7
$# lco or a1    a2     a3     v1     v2     v3
   0.000   0.000   0.000   0.000   0.000   0.000
*END
keyword updatekind
MAT_RIGID
KEYWORD INPUT 3
$ Material : Pusher
```

100

```
*MAT_RIGID_TITLE
Ship
$#    mid      ro      e      pr      n   couple     m    alias
      3 900.00000 9.0000E+9  0.300000   0.000   0.000   0.000
$#   cmo   con1    con2
       0      0      0
$# lco or a1    a2      a3      v1      v2      v3
   0.000   0.000   0.000   0.000   0.000   0.000
*END
keyword updatekind
MAT_RIGID
KEYWORD INPUT 4
$ Material : Steel
*MAT_USER_DEFINED_MATERIAL_MODELS_TITLE
Steel
$#    mid      ro      mt     lmc    nhv  iortho   ibulk     ig
      4 7890.0000      46      16      20      0      4      3
$#  ivect   ifail  itherm  ihyper   ieos    lmca  unused  unused
       1      1      0      0      0      8      0      0
$# p1       p2      p3      p4      p5      p6      p7      p8
2.1E11   0.3     8.0769E10 1.75E11 2.85E8   7.4E8    0.24
$# p1       p2      p3      p4      p5      p6      p7      p8
0.7      1
*END
keyword updatekind
MAT_USER_DEFINED_MATERIAL_MODELS
KEYWORD INPUT 98
*CONTROL_TERMINATION
$# endtim   endcyc   dtmin   endeng   endmas
 0.130000      0   0.000   0.000   0.000
*END
keyword updatekind
CONTROL_TERMINATION
save keyword "C:\Users\Torstein\Documents\1C\testIS.k"
```