Simulation of Ice Rubble Failure against a Conical Structure with Arbitrary Lagrangian-Eulerian Finite Element Method

Janne Ranta, Kari Santaoja, Jukka Tuhkuri

Aalto University School of Science and Technology, Department of Applied Mechanics, Mechanics of Materials
PL 14300, Puumiehenkuja 5 A
00076 Aalto, FINLAND
janne.ranta@tkk.fi

Ice structure interaction process is complex event where different types of ice features together with spatially varying mechanical properties are present. Furthermore, size and shape as well as mechanical properties of the system will change during the process. Due to the versatility of ice-structure interaction events, numerical simulation methods are needed to obtain more understanding about different mechanisms behind field observations.

Arbitrary Lagrangian-Eulerian finite element method based procedure was developed and applied for simulations of ice-structure interaction processes. The procedure was implemented into Abaqus 6.7-1 finite element code. In test simulations, homogeneous consolidated ice rubble field with constant thickness interacted with Kemi-I lighthouse foundation. Constitutive modelling of the ice rubble field was carried out by using linear Drucker-Prager model. Cohesive softening of the material was taken account.

As a result, geometric evolutions, force-time histories as well as some stress data from simulations were obtained. Arbitrary Lagrangian-Eulerian finite element based procedure was found to be a potential method for ice-structure interaction simulations. The procedure allows implementation of sophisticated material models as well as complex ice features in a sense that initial state of the material model is defined differently in different parts of the model.
1. Introduction

The finite element method (FEM) based simulation of ice-structure interaction process is a complex task. Among material, contact and geometric nonlinearities, one has to find out a solution on how to deal with a problem arising from the drastic change in the model geometry during the simulation process. In a typical ice-structure interaction process, the deformation of ice will be far beyond what is seen in a common structural engineering problem. The development of new techniques and computer codes to tackle problems on modelling large deformations and to handle the geometric updates of the model is time consuming. As an alternative approach, one can take a look on existing commercial codes and investigate their applicability on simulation of different processes under interest.

In this paper, a procedure to simulate ice rubble-structure interaction processes is introduced. The procedure is based on the use of arbitrary Lagrangian-Eulerian (ALE) finite element description as it’s implemented in Abaqus software. The basic idea behind the ALE description is that a material and finite element mesh motions are independent on demand. More detailed view on the subject can be found from (Ranta, 2009). The description of the ALE finite element method is given in (Belytschko, 2000), for example. The procedure is discussed in Section 2 by introducing a finite element model with all necessary details. In Section 3 results from analyses i.e. selected force-time histories and geometric evolutions of the rubble piles are presented and failure mechanisms of the ice rubble field are considered.

2. Finite element model

The finite element model consists of three structural parts. For simplicity, only one half of the system is considered. The two most important parts are the ice rubble field and the structure. In addition, a vertical plane is used to provide a symmetry boundary condition for the ice rubble field behind the structure. The overall view of the finite element model and detailed dimensions of the structure are shown in Figure 1.

Both the structure and the vertical wall are modelled as analytical rigid surfaces. The ice rubble field is a deformable body with a uniform thickness (4 m). The dimensions for the Kemi-I lighthouse foundation are adopted from the article (Brown and Määttänen, 2009).
2.1 Mesh design

To be able to model changes in the ice geometry, one has to be able to update the finite element mesh repeatedly to avoid element distortions and numerical errors. One way to take care of the model update is to apply the arbitrary Lagrangian-Eulerian finite element method with adaptive constraint planes. In this method, the topology of the model discretization will remain unchanged (nodal connectivity do not change). Therefore adaptive constraint planes are needed to keep the finite element discretization around the structure as a constant like feature, particularly in the horizontal direction. The use of adaptive constraint planes makes it also possible to have a denser finite element mesh near the structure (Figure 2).

![Figure 2. Ice-structure interaction model and adaptive constraint planes cp1, cp2 and cp3. Velocity type of boundary conditions on rubble field boundaries are shown as well.](image)

In the present model, the structure will move towards the ice rubble field by a constant velocity $v_s$. Therefore, to keep the dense finite element mesh around the structure, the adaptive constraint planes must follow the structure. Adaptive constraint plane movements can be performed at discrete points of time. It follows that the arbitrary Lagrangian-Eulerian finite element description is needed only when the adaptive constraint planes are moved. Thus, in practice the whole analysis can progress by traditional Lagrangian finite element description, because only short time is needed for each movement of adaptive constraint planes.

2.2 Gravity, buoyancy and balancing pressure

Gravity and gravity based loads, e.g. buoyancy, need to be included in the model. This can be done by using a body load vector field $\mathbf{p} = \mathbf{p}(z)$:

$$\mathbf{p} = \begin{cases} (\rho_w - \rho_l)(1 - \eta) \mathbf{g} \mathbf{e}_3, & z < s \\ \rho_l (1 - \eta) \mathbf{g} \mathbf{e}_3, & z \geq s \end{cases}$$

[1]
where parameters $\rho_w$, $\rho_i$ and $\eta$ are water density, density of ice and porosity of the ice rubble, respectively. The parameter $s$ defines the distance from the $x$-$y$ plane of the global coordinate system to the sea level and the $g$ is the gravitational acceleration. Coordinate $z$ is measured from the $x$-$y$ plane to the direction of the base vector $\hat{e}_3$ that is pointing towards the sky.

In Abaqus, ALE finite element technology is available with C3D8R elements. C3D8R element is a linear eight noded element with a single integration point only. Calculation of body load is performed according to the location of the so called load definition point that in the case of C3D8R element coincides the integration point (the center of mass) of an element. As a consequence of the use of single load definition point for each volume element, buoyancy modelling error will be present in the case of partially submerged elements. For large elements, the modelling error can be reduced by using the concept of balancing pressure discussed in (Ranta, 2009) and illustrated in Figure 3.

![Figure 3. The concept of balancing pressure $p_c$ used to reduce the effect of incorrect buoyancy caused by partially submerged elements with single load definition point.](image)

The balancing pressure can be expressed as a function

$$p_c = \rho_w (1-\eta) g \zeta , \ 0 \leq \zeta \leq \frac{1}{2} h_e,$$

where $h_e$ is the height of submerged elements and $\zeta$ is the measure from the top surface of the ice rubble field to the water line.

### 2.3 Stepwise solution procedure

In Abaqus explicit, time and spatially varying loads can be included in by using user defined subroutine called VDLOAD. Subroutine based body loads and ALE finite element technology are not compatible functions in Abaqus. Therefore, a stepwise solution procedure needs to be applied. In the stepwise solution procedure the simulation is divided into two different types of analysis steps: pure Lagrangian formulation and ALE formulation based steps. During the pure Lagrangian steps the system is fully defined with all desirable physics included in. On the contrary, during the ALE steps all VDLOAD based body and surface loads need to be excluded due to the incompatibility issue.

The purpose of the ALE formulation based steps is to carry out the mesh movements in a way that the dense mesh region follows the structure. In ALE finite element technology, the mesh
movements can be performed without any actions to material particle motions. The finite element mesh itself is massless and therefore no inertial forces are generated. A problem is that gravitational loads are removed from the ALE steps and therefore these steps are mathematically but not physically correct. Anyway, in order to limit this unphysical behaviour, the duration $t_A$ for the ALE steps can be chosen to be very small. Actually, the desired movements of the nodes of the finite element mesh can be performed within a single computation increment and therefore $t_A = 2 \times 10^{-5}$ s was used. The duration $t_L$ for the pure Lagrangian formulation based steps needs to be chosen in a way that severe element distortions are not faced before the next ALE step and mesh improvement take place.

### 2.4 ALE functions

In Abaqus, ALE adaptive meshes can be controlled by several functions. From the point of view of this work, the most important thing is the applicability to define adaptive constraints that can be used to control mesh motions independently from the material motions. Adaptive constraints are applied to move or restrict the movements of nodes belonging to constraint planes depicted in Figure 2 for example. In fact, constraint planes $cp1$ and $cp2$ are set to follow the structure in a way that within each ALE adaptive step, velocities $v_{c1}$ and $v_{c2}$ are fixed according to the equation

$$v_{c1} = v_{c2} = \frac{t_L}{t_A} v_s.$$  \[3\]

The velocity $v_{c3}$ for the adaptive constraint plane $cp3$ needs to be set to zero to retain the dense mesh region close to the structure. Since the duration $t_A$ for the ALE formulation based analysis steps is much smaller than the duration for the Lagrangian formulation based analysis steps, velocities $v_{c1}$ and $v_{c2}$ become very large. Under the fast translation of constraint planes $cp1$ and $cp2$, one needs to maintain some mesh control parameters related to a mesh smoothing to have decently improved mesh. Such parameters define the mesh smoothing technique and how many times the mesh smoothing algorithm will be called at a time.

### 2.5 Material model

A material model is one of the key factors in modelling ice-structure interaction processes. For ice rubble, a cohesive frictional behaviour is often assumed, and various analytical formulas to compute ice rubble loads are based on the classical Mohr-Coulomb yield criteria, see e.g. (Timco & al. 2000). In numerical models, Drucker–Prager yield criteria and its extended or modified Drucker–Prager –cap versions can be applied (Figure 4).

![Figure 4. Different versions of the Drucker-Prager type yield criteria available in Abaqus. (a) linear, (b) extended and (c) modified Drucker-Prager cap -model.](image-url)
In Abaqus, a user can define own material models. At the moment, one such kind of a model for ice rubble-structure interaction modelling is a shear–cap model discussed in (Heinonen, 2004). For simplicity, the linear version of the Drucker–Prager material model is applied, although it is known that it is incapable of taking into account compaction of a material due to hydrostatic pressure, for example. The linear Drucker–Prager yield criteria can be expressed by means of hydrostatic pressure stress $p$, equivalent von Mises stress $q$, friction angle $\beta$ and the cohesion $d$. The Drucker-Prager yield function is

$$f(\sigma) = q - p \tan \beta - d = 0.$$ \[4\]

Yielding of the Drucker–Prager material is controlled by the flow potential function $g(\sigma)$ that can be obtained by replacing the friction angle $\beta$ with a dilatation angle $\psi$ in equation 4. Another drawback of the Drucker–Prager material model is that with high values of $\psi$ the material may experience unrealistic volumetric expansion. In practise, measured values for the friction angle $\beta$ can be high and therefore, when one uses associative flow rule ($\psi = \beta$), the dilatant behaviour of the material may become unrealistic. To have more control over the dilatant behaviour of the system, one can use non-associative flow potential function with low dilatation angle or to choose a material model equipped with a cap yield surface for example.

Softening of the ice rubble is primarily caused by the failing of freeze bonds between ice blocks. In addition, other failure mechanisms, e.g. crushing of the ice, related to global ice rubble deformations can be identified (Heinonen, 2004). According to Heinonen (2004), the cohesive softening law for the ice rubble can be given as a function of equivalent plastic strain $\varepsilon^{pl}$. In the present simulations, a slightly modified version of the softening law is applied and it can be expressed in a form

$$d = (d_0 - d_f) \exp \left[-\frac{\varepsilon^{pl}}{\kappa}\right] + d_f. \quad [5]$$

In equation 5, parameters $d_0$, $d_f$ and $\kappa$ are initial cohesion, final cohesion and softening parameter respectively. The final cohesion is limited such that $0 \leq d_f < d_0$. The original form of the softening law used by Heinonen (2004) is obtained if $d_f = 0$.

### 2.6 Contact modelling

There is a need to model contacts between the ice rubble field and the structure. As a benefit of the modelling technique, only a relatively small portion of the model needs to be covered by contact searches. Contact definitions are created for contacts between the dense mesh region of the ice rubble field and the surface of the structure. Contact modelling is carried out by using surface-to-surface contact model with kinematic mechanical constraint formulation. Hard contact model is applied for direct contacts whereas penalty friction formulation is used for the contacts in tangential direction. The same contact definitions are valid in both pure Lagrangian and arbitrary Lagrangian–Eulerian analysis steps. Under these contact definitions, the only contact property input in Abaqus is for friction coefficient $\mu$ covering both static and dynamic cases for non-moving and moving interaction surfaces respectively.
3. Results

The modelling technique introduced above was applied by conducting seven Abaqus runs. Approximately 45,000 C3D8R elements were used to construct the finite element mesh for the whole model. Inside the dense mesh region, fourteen elements were used in the thickness direction of the rubble. On the contrary, inside the coarse mesh region four elements in the thickness direction were used only. In a single simulation, altogether 121 analysis steps were used to obtain approximately a 15.25 m penetration of the structure. The odd numbered steps (61/121) were based on the Lagrangian finite element formulation whereas the even numbered steps (60/121) were based on the ALE finite element formulation. The stable time increment size of each Lagrangian steps was left to be chosen by Abaqus. The parameters related to the analyses are collected into Table 1.

Table 1. Parameters applied in Abaqus analyses.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E$</td>
<td>100</td>
<td>MPa</td>
<td>Young’s modulus</td>
</tr>
<tr>
<td>$\nu$</td>
<td>0.3</td>
<td></td>
<td>Poisson’s ratio</td>
</tr>
<tr>
<td>$g$</td>
<td>9.81</td>
<td>m/s²</td>
<td>Gravitational acceleration</td>
</tr>
<tr>
<td>$\rho_w$</td>
<td>1000</td>
<td>kg/m³</td>
<td>Density of water</td>
</tr>
<tr>
<td>$\rho_i$</td>
<td>920</td>
<td>kg/m³</td>
<td>Density of ice</td>
</tr>
<tr>
<td>$\eta$</td>
<td>0.3</td>
<td></td>
<td>Porosity of ice rubble</td>
</tr>
<tr>
<td>$d_0$</td>
<td>8…25</td>
<td>kPa</td>
<td>Initial cohesion</td>
</tr>
<tr>
<td>$d_f$</td>
<td>1</td>
<td>kPa</td>
<td>Final cohesion</td>
</tr>
<tr>
<td>$\kappa$</td>
<td>0.03</td>
<td></td>
<td>Softening parameter</td>
</tr>
<tr>
<td>$\mu$</td>
<td>0.025</td>
<td></td>
<td>Friction coefficient</td>
</tr>
<tr>
<td>$v_s$</td>
<td>0.25</td>
<td>m/s</td>
<td>Velocity of the structure</td>
</tr>
<tr>
<td>$t_L$</td>
<td>0.5</td>
<td>s</td>
<td>Lagrangian step duration</td>
</tr>
<tr>
<td>$t_A$</td>
<td>2</td>
<td>μs</td>
<td>ALE step duration</td>
</tr>
<tr>
<td>$v_{c1}$ &amp; $v_{c2}$</td>
<td>0.625</td>
<td>km/s</td>
<td>Constrained velocity of cp1/cp2</td>
</tr>
<tr>
<td>$v_{c3}$</td>
<td>0</td>
<td>m/s</td>
<td>Constrained velocity of cp3</td>
</tr>
<tr>
<td>-</td>
<td>0.8, 0, 0.2</td>
<td></td>
<td>Weights for combining mesh smoothing methods</td>
</tr>
<tr>
<td>-</td>
<td>10</td>
<td></td>
<td>Mesh sweeps per increment</td>
</tr>
</tbody>
</table>

Seven analyses were performed in two analysis series, namely in series A and series B. In series A, four analyses were carried out by using the ideal plasticity theory, where the cohesion of the material remains unchanged ($d = d_0$ and $d_f = 0$). In series B, three analyses were conducted. The case B1 was similar to case A1, but instead of associative flow rule ($\psi = \beta$), the dilatation angle $\psi = 15$ degrees was applied. The reduction of the dilatation angle was made for having a clear picture on the volumetric behaviour of the applied material. Otherwise the associative flow rule was used. Analyses B2 and B3 were similar to analyses A2 and A3, but cohesive softening was taken account. In general, initial cohesion $d_0$ and the friction angle $\beta$ were varied according to Table 2. The parameters shown in Table 2 were chosen according to Heinonen (2004, Figure 7.14).

From each simulation run, four types of quantities including energy, stress, force and geometric data were recorded. In all analyses, the sampling rate was 42 Hz. In addition, data points related to the short ALE formulation based steps are left out.
Table 2. Used parameter combinations for initial cohesion $d_0$ and friction angle $\beta$. Both Mohr-Coulomb (MC) and Drucker-Prager (DP) parameters are presented.

<table>
<thead>
<tr>
<th>Cohesion [kPa]</th>
<th>Friction angle [deg]</th>
</tr>
</thead>
<tbody>
<tr>
<td>MC 5.8</td>
<td>DP 19.5</td>
</tr>
<tr>
<td></td>
<td>MC 10</td>
</tr>
<tr>
<td>4.9</td>
<td>8</td>
</tr>
<tr>
<td>7.3</td>
<td>12.5</td>
</tr>
<tr>
<td>10.7</td>
<td>17.5</td>
</tr>
<tr>
<td>14.5</td>
<td>25</td>
</tr>
</tbody>
</table>

| Cohesion [kPa] | MC 8                  |
|               | DP -                  |
| 4.9           |                      |
| 7.3           | A2, B2               |
| 10.7          | A3, B3               |
| 14.5          | A4, B4               |

*In B1, $\psi = \frac{1}{2} \beta$

3.1 Force-time histories

Force-time histories in $x$- and $z$-directions are shown in Figures 5 and 6. Despite the use of the rough stepwise solution procedure, all force curves are having a clearly continuous trend. Force peaks that can be seen in Figures 5 and 6 are caused by the use of the stepwise solution procedure where the Lagrangian formulation based analysis step is suddenly changed to ALE formulation based step and vice versa.

In Figure 5, total reaction forces (half model forces multiplied by a factor two) recorded in $x$-axis directions are shown. Force curves related to analysis series A are arising much more aggressively compared with corresponding analysis series B results. In analysis series A, the material was ideally plastic whereas in the analysis series B the cohesive softening (excluding B1) was taken account. In Figure 6, total reaction forces in $z$-axis direction are shown. From Figures 5 and 6 it can be seen that peak loads related to the performed analyses were not achieved. In the case of ideal plasticity theory, all force curves are arising almost monotonically. On the contrary, in models where cohesive softening has been taken account, a clear drop down at force levels can be observed after the first peak load. In the case of ideal plasticity theory, the material won’t lose its load carrying capacity and ice flow around the structure can’t happen with the same ease as with models where the cohesive softening will take place. Resisted ice flow makes the rubble pile in front of the structure to grow faster and therefore also the recorded loads become higher.

From Figure 5, one can observe that maximum load related to the case B3 is obtained approximately at time 18 s. This load is also higher than the load peak seen in the beginning of the same analysis. The first load peak can be predicted by using analytical ice load calculation methods, but to have prediction on the load levels at time 18 s for example, one has to be able to perform process simulations that take account changes in the model geometry and in the state of the applied material as well.

In analyses, where the cohesive softening was taken account, the top surface of the ice rubble field folded in a way that reasonable continuation of analyses would have require the use of rubble to rubble contact modelling. Therefore the analysis time was limited to 30 s in all analyses except analysis B2 where the folding took place at time 25 s. In analyses A1, A2, A3, A4 and B1, where the ideal plastic material was applied, the contact forces became unreasonable high and no peak load like in the case B3 in Figure 5 was found.
3.2 The geometric evolution of ice rubble piles

The geometric evolution of rubble piles was investigated by recording the locations of the highest and lowest point of each pile. The coordinates shown in Figure 7 are not exactly from the highest and lowest points of rubble piles due to use of ALE finite element description and ALE adaptive constraint planes. This can be seen from Figure 7 and on the curve describing the lowest point of case A1, for example. The decrease in rubble depth after 24 s is not physical, but is caused by mesh updating and transfer of nodes. Anyway, coordinates are believed to be well representative most of the time during analyses. From the left hand side of Figure 7, one can see the difference between analyses A1 and B1 where only the dilatation angle was varied. The 50% reduction of the dilation angle has caused approximately 20-25% reduction in the rubble pile height. This difference has clear effects on force curves shown in Figures 5 and 6 too.

**Figure 5.** Total reaction forces in x-direction measured from the structure. Half model forces are multiplied by a factor two.

**Figure 6.** Total z-direction reaction forces measured from the structure. Half model forces are multiplied by a factor two.
In addition, rubble pile evolutions during cases A3 and B3 are compared. These comparisons are disclosed in Figure 8. On the left hand side of Figure 8, side views of rubble piles are shown. Different colours represent the variations of the equivalent plastic strain. In turn, on the right hand side of the Figure 8, rubble pile contours are shown. Contours are describing rubble pile geometries above the sea level only. The widest contours (highlighted with small red dots) are lying approximately at the level of the freeboard (FB) of the ice rubble field. Vertical distance between adjacent contour lines is 0.5 m. In Abaqus, when the large deformation theory is activated, an incremental strain itself is computed as a natural logarithm of an increment of the left stretch tensor.

![Figure 7. Locations of the highest and lowest points of rubble piles.](image)

![Figure 8. Rubble pile comparisons between analyses A3 and B3. Side views of rubble piles on the left and rubble pile contour plots on the right.](image)
In analysis series B, the ice rubble failure modes shown in Figure 9 were found. At the beginning of analyses, until the first force peaks were fully developed, an inclined narrow plastic zone, slip plane, through the rubble field was found. Afterwards, the inclined slip plane turned to be a combination of two inclined planes between intact and softened material regions for both upper and lower parts of the ice rubble field.

![Figure 9. At first, the rubble field failed by a single failure plane (on the left). Afterwards, the failed zone of the ice rubble was expanded by a V-shaped failure plane (on the right). In the grey region, the plastic strain is zero – the ice rubble has not failed.](image)

4. Discussion

Comparison of the elastic reaction forces related to the first force peaks recorded in x-axis direction in cases A1/B1 (0.62 MN), A2/B2 (0.64 MN) and A3/B3 (1.05 MN) show quite good agreement with keel loads of 0.7 MN and 0.9 MN reported by Brown and Määttänen (2009). Reported keel loads were separated from the ridge loads of 2.1 MN and 4.0 MN respectively. The first force peaks were used in comparison because ice loads are believed to be too high in analyses where ideal plastic material was used. On the contrary, in models where cohesive softening was taken account, the highest load was not much larger than what was predicted by the by the corresponding first force peak, see e.g. Figure 5. On the other hand, the failure modes of the ice rubble field shown in the Figure 9 are in good correspondence with the assumptions of failure planes behind many analytical ice load calculation algorithms, like the one described in (Mellor, 1980). Nevertheless, the characteristics of the obtained rubble pile geometries have not been compared with those what have been observed in reality.

A procedure for ice rubble-structure interaction simulations was developed and successfully applied. Further development of details of the modelling technique is needed to have more reliable results and to be able to simulate ice–structure interaction processes related to different kinds of ice features, e.g. ice ridges. One important thing is to tackle problems related to the stepwise solution procedure that causes the force peaks seen in all force-time histories. In Abaqus, ALE finite element description accepts user defined materials (VUMAT subroutine) and therefore e.g. implementation of simple ice ridges with spatially varying strength is possible by setting spatially varying initial state for the applied material.

Acknowledgements

The Abaqus licence and software was provided by the Finnish IT–center for science. This work has been conducted during the AHRAVUO (Ridge-structure interaction simulation) project. The project sponsors ABB Marine, Aker Arctic Technology Inc., Fortum, Finnish Maritime Administration, Tekes (The Finnish Funding Agency for Technology and Innovation) and VTT (Technical Research Centre of Finland) are greatly acknowledged.
References


