Experimental and Numerical Study on the Impact Strength of Beams and Plates

Bin Liu
Centre for Marine Technology and Engineering (CENTEC), Technical University of Lisbon, Instituto Superior Técnico, Lisboa, Portugal

ABSTRACT: The paper aims at summarizing the assumptions adopted in numerical models of small-scale ship structural elements subjected to lateral impact. The calculations are carried out using the finite element package LS-DYNA. The assumptions adopted in the numerical model are validated by means of drop weight impact tests. The study considers the following ship structural components: beams, plates and stiffeners with attached plate. The impact behavior of the structural elements is validated by comparison of their experimental and numerical force-displacement responses. Special attention is paid in the definition of the true stress-strain relationship and in the representation of the experimental boundary conditions.

1 INTRODUCTION

Ship grounding and collision could cause loss of human lives and severe environmental damage. In order to minimize these consequences, it is necessary to design crashworthy marine structures. The structural design of ships concerning collision requires an accurate prediction of the extent of damage of ship structural components under impact loading. Finite element analysis is a useful tool to predict this extent of damage. However, the nonlinear dynamic analysis should be compared with experimental tests before being used for structural design.

Theoretical and experimental analyses of individual ship structural components under lateral impact loads, such as beams and plates, have been widely investigated. However, comparison between experiments and numerical simulations still require investigation especially when fracture occurs. Thus, this paper aims at summarizing assumptions adopted in numerical models of ship structures under lateral impact load which are compared with experimental impact tests until maximum load and fracture.

Three types of ship structural components were studied: beam, plate and stiffened plate. Concerning the response of beams; the studies have been focus on their plastic response when stuck by a mass along the span, defining the boundaries of the striking mass and the beam, and also their failure modes. Parkes (1955) studied the plastic deformation of beams struck by a mass, summarizing the influence of the boundary conditions of the striking mass in the beams dynamic response. Parkes (1958) investigated the permanent deformation of clamped beam struck transversely along its span by striking mass. Jones (1973) conducted a theoretical study to examine the influence of axial displacement at the supports. Menkes and Opat (1973) proposed three basic failure modes for fully clamped beams: large inelastic deformation, tensile tearing and transverse shear failure at the supports. Chen and Yu (2004) investigated the failure behavior of clamped beams with one or two notches under impact loading. Villavicencio and Guedes Soares (2009) summarized the plastic response of clamped beam impacted along its span and compared the experimental results with a theoretical rigid plastic analysis and with finite elements. Villavicencio and Guedes Soares (2011a) presented a numerical model to simulate the experimental boundary conditions of beam impacted transversely by a mass along its span.

Similar aspects have been summarized for the response of plates under impact loads. For example, Cox and Morland (1959) gave the theoretical solution for a simply supported square plate subjected to a uniformly distributed rectangular pressure pulse. Jones (1971) proposed an approximate theoretical procedure to estimate the permanent transverse deflections of rectangular plates under uniformly distributed loading. Yu and Chen (1992) completed a theoretical investigation to trace the plastic response of simply-supported or fully-clamped rectangular plates. Shen et al. (2002) presented a series of tests to examine the plastic response and failure of thin circular plates struck transversely at the centre by a mass. Jones et al. (2008) studied the perforation of steel square and rectangular plates struck by different cylindrical striking mass.

Previous studies on stiffened plates are mainly orientated to response of ship side and bottom structures. Manolakos and Mamalis (1985) used a rigid plastic analysis for predicting the impact response of stiffened plates. Ehlers et al. (2008) performed numerical simulations of the impact response of ship side structures, finding a strong influence of the adopted failure criterion. Cho and Lee (2009) developed a simplified method for the prediction of the extent of damage on stiffened plates subjected to lateral impact. Alsos et al. (2009) investigated two failure criteria which were implemented into the impact
analysis, studying the influence of element size on failure. Villavicencio and Guedes Soares (2011b) studied numerically the deflection and failure of small panels subjected to lateral impact using different stiffener distributions and impact locations.

This paper aims at summarizing the assumptions adopted in numerical models of small-scale ship structural elements subjected to lateral impact. The experimental and numerical methods are used to study the impact resistance of structural components. Detailed information of the impact response of structural components is obtained through drop weight impact tests and nonlinear finite element simulations. The study considers the following ship structural components: beams, plates and stiffeners with attached plate. The impact behavior of the structural elements is validated by comparison of their experimental and numerical force-displacement responses. Special attention is paid in the definition of the true stress-strain relationship and in the representation of the experimental boundary conditions.

2 THEORETICAL BACKGROUND

The nonlinear finite element method is one of the most powerful approaches to simulate ship collisions and groundings. However, the results of finite element simulations are significantly affected by the modeling technique used. In this paper, the theoretical background for nonlinear finite element simulations includes material stress-strain relationship, mesh size, dynamic yield strength, failure criteria and contact-impact definition.

Since the material information obtained from the engineering stress-strain curve is only valid to obtain true stress-strain relationship until necking, mathematical approximations are used to define the true curve beyond maximum load. Thus, beyond necking two approximations are used: the simple power law relation and the true material curve proposed by Zhang et al. (2004).

In finite element simulations, the mesh size has important effects on the results. Thus, to capture highly nonlinear characteristics of structures, a very fine finite element mesh is required. However, the mesh size is not very important in analysis of plastic deformation, but plays an important role when fracture occurs.

The yield stress of material is dependent on both strain rate and the temperature at which the deformation occurs. Thus, the relationship between yield stress and strain rate at constant temperature can be expressed by the dynamic yield strength of material. The strain rate sensitivity function is often given using the Cowper–Symonds equation (Cowper and Symonds 1957) in which the coefficients depend on the material.

Usually the first rupture of an element is defined by the critical failure strain value. The elements are deleted in the finite element model when they yield this critical value, which is strongly dependent on the mesh size. The tensile test simulation can be used to define the critical failure strain and mesh size, compared with the engineering stress-strain curve.

The treatments of sliding and impact interfaces are important definitions in nonlinear analyses when two bodies interact (Hallquist 2010). Interfaces can be defined in three dimensions by triangular and quadrilateral segments. One side of the interface is designated as the slave side, and the other as the master side.

The above definitions are very important in the nonlinear finite element analysis of structures subjected to impact loads. Consequently, good agreement is obtained between experimental and numerical results when those parameters are defined appropriately in numerical models.

3 PLASTIC RESPONSE OF BEAMS

Experiments and numerical simulations were conducted in order to study the plastic response of beams laterally impacted by a mass. Most of researches use standard boundary conditions, such as simple supported and fully clamped. However, studies on the influence of the coefficient of friction at the supports are rare. Thus, this section studies the plastic response of beams using the definition of the coefficient of friction at the modeled experimental supports.

3.1 Experimental details

The impact tests represent a situation in which it is considered that a partially clamped beam is struck at the mid-span by a mass travelling with an initial impact velocity, as shown in Figure 1. The beam is partially clamped between two support plates, i.e. the beam experiences some axial displacement at the supports during the impact event.

Figure 1 Beam struck transversely by a mass.

The experimental set-up can be seen in Figure 2. The specimen beams were 125 mm span length, 20 mm width and 3 mm thickness. The length of the beams is divided in three: span length and two support lengths, where the span length is the distance between the supports and the support length is the...
length of the upper support plate. Tests were carried out by using a striking mass of 24.42 kg and the impact velocities vary from 0.5 to 2.5 m/s.

3.2 Experimental results

The experimental results of the tested beams are summarized in Table 1. It is observed that the force and the displacement are in direct proportion to the impact velocity. As is a drop weight impact experiment, the maximum absorbed energy during the impact is larger than the initial input kinetic energy of the striking mass. Hence, the total absorbed energy includes the potential energy of striking mass caused by the vertical deformation of the beam.

<table>
<thead>
<tr>
<th>Impact Velocity (m/s)</th>
<th>Values at Peak Force</th>
<th>Values at End</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Force (kN)</td>
<td>Displ (mm)</td>
</tr>
<tr>
<td>0.5</td>
<td>1.60</td>
<td>3.65</td>
</tr>
<tr>
<td>1.0</td>
<td>2.86</td>
<td>7.37</td>
</tr>
<tr>
<td>1.5</td>
<td>4.27</td>
<td>12.64</td>
</tr>
<tr>
<td>2.0</td>
<td>5.85</td>
<td>16.64</td>
</tr>
<tr>
<td>2.5</td>
<td>8.35</td>
<td>21.99</td>
</tr>
</tbody>
</table>

3.3 Numerical model

The numerical model is illustrated in Figure 3. The specimen beam and the striking mass were modeled with solid (8-nodes, 1-integration point) elements, using mesh sizes of 2×2×1.5 mm and 1×1×1 mm.

The material selected from the library of LS-DYNA was ‘Mat.024-Piecewice lineal plasticity’, which allows the definition of a true stress-strain curve as an offset table. The true material uses the exact true stress-strain relationship until maximum load and the approximate relationship proposed by Zhang et al. (2004) beyond necking.

The material of striking mass was defined as ‘Mat.020-Rigid’. As the falling weight assembly was modeled as a simple box, an artificially large density was used to give the same mass as used in the experiments. In the striking mass, only the vertical translation was free, in which direction the initial impact velocity was assigned. The “Load Body Z” was chosen to define the acceleration of gravity.

The support plates were modeled with 4-node shell elements, defining the same rigid material of the striking mass. All the degrees of freedom of the lower support plates were constrained, whereas only the vertical translation was free on the upper support plates. The “Boundary Prescribed Motion Node” was selected to define preload force of bolts on the upper support plates, and the preload force was 37500 N.

The contact between the striking mass and the beam, and the contact between the support plates and the beam, was defined as “Automatic Surface to Surface”.

3.4 Numerical results

The tested beam with an impact velocity of 1.0 m/s was selected to compare with the numerical simulations. The sensitivity of the coefficient of friction between the beam and the support plates was evaluated by modifying its magnitude in the contact definition. The results of the simulations are shown in Figure 4.

Several coefficients of friction were chosen to analyze the influence on the plastic responses of beam. The result of numerical simulation with coefficient of friction of 0.21 gives better agreement with the experimental result.

The maximum force and displacement are shown in Figure 5. The numerical results show a straight line, predicting well the maximum displacement in the full range of impact velocities. However, the impact forces tend to deviate at higher impact velocities.
Figure 5 Maximum force and displacements at different impact velocities.

The axial displacements at the supports were measured at the end of the impact event in order to analyze the coefficient of friction. The comparison of the axial displacements at the supports is shown in Figure 6. The axial displacements of the simulations show good agreement with the experimental results.

Figure 6 Axial displacements at the supports at different velocity.

4 FAILURE PREDICTION OF PRE-NOTCHED BEAMS

In this section, experiments and numerical simulations were conducted to study the plastic behavior and fracture propagation of clamped pre-notched beams stuck laterally by a mass. Beams with and without notches were tested, considering different notches and impact points. The material definition in the numerical model of impact test was obtained from numerical result of tensile test.

4.1 Experimental details

The impact tests are similar to the previous cases. The specimen beams were 250 mm span length, 20 mm width and 8.0 mm thickness. Beams with and without notch were tested. Two depth of notch were considered: 2.0 mm and 4.0 mm. The width of the notches is 1.0 mm. The positions of the notch were 15, 75 and 115 mm from the support. Tests were carried out using a striking mass of 74.8 kg and an impact velocity of 4.2 m/s. The indenter was a hemispherically ended cylindrical projectile of diameter 30 mm. The impact point was at the mid-span and at one-quarter of the support.

4.2 Experimental results

The time dependant curves of force, displacement and absorbed energy and the force-displacement response are shown in Figure 7. The specimen N15_2mm_M experienced large plastic deformation, whereas the specimen N15_4mm_M fractured at the notch location. Both curves are similar at the beginning of the response, indicating that the depth of notch has small influence on the plastic response of pre-notched beams. The absorbed energy until fracture is the most important parameter of the impact response of pre-notched beam.

Figure 7 Experimental results of force-time, displacement-time, absorbed energy-time and force-displacement. P: Specimen N15_2mm_M (plastic deformation); F: Specimen N15_4mm_M (fracture). N15: Notch position 15 mm from the support; 2mm and 4mm: The depth of notch 2.0 mm and 4.0 mm, respectively; M: Impact at mid-span.

4.3 Numerical model

The specimen beam and the striking mass were modeled with solid elements, and with mesh size of
1×1×1 mm. In the numerical model, the beam length was the span length, and all the degrees of freedom of the ends of beam were constrained. The definition of striking mass is the same with Section 3.3.

4.3.1 Material definition
The tensile test was modeled in LS-DYNA in order to verify the plastic response of the material and to explore the stress and strain distribution in the neck at failure. The tensile tests used to obtain the mechanical properties of the material were numerically simulated in order to predict the plastic response of the impact specimen and the critical failure strain until fracture. The engineering and true stress-strain material curves are plotted in Figure 8. In the present work, the material curve defined by Zhang (2004) is denoted by “GL”, the power law curve is denoted by “PL” and the true material defined until necking and continued with the mathematical expression of Zhang (2004) beyond localization is denoted by “UN+GL”.

![Figure 8 True and engineering material curves.](image)

The size of model is 100×20×8 mm, and the mesh size is 1.0 mm. The critical failure strain (\(\varepsilon_f = 1.2\)) was obtained by successive numerical simulation. Figure 9 shows the results of the numerical simulations using the three material curves. The numerical results of GL and UN+GL material curve are far from the engineering stress-strain curve, and the numerical result of PL material curve is the best approximation, which follows the experimental curve quite precisely.

![Figure 9 Results of numerical simulations.](image)

4.4 Numerical results
The true stress-strain curve (PL), mesh size (1.0 mm) and failure strain (1.2) were determined by the numerical simulation of the tensile test. Two impact specimens were selected for the numerical simulations. These beams have two different depths of notch, but they are located at the same position which is near the support. Both of them were impacted at the mid-span. One beam sustained large plastic deformation and the other was fracture at the notch position. The experimental and numerical force-displacement curves are plotted in Figure 10. The results are similar with the experiments. The cause of the difference is that the boundary condition in the experiments is not absolutely fully clamped, but the experimental result is still reasonable. The appropriate true stress-strain curve and critical failure strain obtained from the simulation of the tensile test, helped to reproduce the experimental response in the impact model.

![Figure 10 Comparison of experimental and numerical force-displacement responses: (a) Specimen N15_2mm_M; (b) Specimen N15_4mm_M.](image)

5 PLASTIC RESPONSE OF RECTANGULAR PLATES
In this section, experimental and numerical analyses of laterally loaded rectangular plate are summarized. The impact tests use different types of indenter to study the influence of shape of indenter on the plastic response of rectangular plate. Also, two plates are impacted to study the force-displacement responses of rectangular plate under different impact velocity.
5.1 Experimental details

The impact tests represent that a fully clamped rectangular plate is struck at the center by a mass. The plate thicknesses are 1.4 and 4.0 mm (henceforth referred to as ‘thin’ and ‘thick’, respectively) in the experiments. Specimen plates were fully clamped by four bolts between two thick rectangular steel plates with internal cut-out of 127 × 76.2 mm. A hemispherical indenter of Ø 30 mm with striking mass 54.34 kg was used to impact two thin plates at velocity 2.7 and 3.3 m/s. The thick plates used six different types of indenters with an impact velocity of 1.94 m/s. The indenters were hemi-spherically ended projectiles of diameter 10, 20 and 30 mm, and cylindrically ended projectiles of diameter 10, 20 and 30 mm.

5.2 Experimental results

The force-displacement responses of thin plates at different velocity are shown in Figure 11. The impacted plates experienced large plastic deformation, and the force-displacement responses are similar at the beginning of the response. The differences just depend on the incident energy.

Figure 11 Force-displacement responses of thin plates at different velocity.

Figure 12 shows the force-displacement responses of thick plates using different types of indenters. It was observed that the shape of the indenter has strong influence on the impact response. All the plates experience large plastic deformation, and the maximum energies are basically the same. The displacement increases and the force decreases when using the smaller diameter of indenter, and this effect is more evident with the cylindrical indenters. The force is larger using the cylindrical indenter and consequently reproduced smaller displacements. This effect is produced because the cylindrical indenter has more contact area and thus indentation out of the plane of the plate is less evident.

5.3 Numerical model

The plate was modeled in 4-node shell elements with 5-integration points through the thickness. In order to check the accuracy, two definitions of the boundary condition were evaluated in the numerical model. The clamped and supported boundary conditions are shown in Figure 13. In the clamped model, the edges of plate were constrained in all degrees of freedom whereas in the supported model, the nodes of the plate at the bolts position were fully clamped, defining implicit contact between the support plates and the specimen plate. The definition of striking mass is the same with Section 3.3.

Figure 13 Boundary conditions of rectangular plate.

5.4 Numerical results

The simulations were evaluated in terms of true stress-strain curve, mesh size, element type and boundary condition.
5.4.1 Thin plates

The experimental results of the thin plates at impact velocity of 2.7 m/s were used to compare the different parameters studied in the numerical model. Since the three evaluated true stress-strain curve (GL, UN+GL, PL) are similar, the impact responses were almost the same. The influence of the mesh size (4.0, 2.0 and 1.0 mm) is very small, and thus the impact model reproduces similar plastic response. Similar results were obtained using the shell and solid elements and the same mesh size 2.0 mm. The comparison between the clamped and supported models is shown in Figure 14. The supported model reproduced higher displacement, but smaller force in the event, because of the sliding between the supports.

![Figure 14 Comparison of different support.](image)

The material strain rate sensitivity is evaluated in the supported model using the coefficient of the Cowper and Symonds constitutive equation for mild and high tensile steel (Figure 15). Using the mild steel coefficients (C=40.4 and q=5), the force-displacement response shows a stiffer behavior. However, using the high tensile steel coefficients (C=3200 and q=5), good agreement with the experiments is noticed at the first stage of impact although the results deviate at the end, leading in higher forces and consequently smaller displacement.

![Figure 15 Comparison of different dynamic yield strength. C40.4q5: mild steel coefficients; C3200q5: high tensile steel coefficients.](image)

5.4.2 Thick plates

As mentioned the different indenters influenced the impact response of plate. Since similar results between the clamped and supported models were obtained, the clamped model was selected for the remaining numerical simulations.

The cylindrical and spherical indenters of Ø30 mm were selected to compare the experimental force-displacement responses (Figure 16). Good agreement between them was found, being the maximum forces and displacements well predicted.

![Figure 16 Force-displacement responses of experimental results and numerical results. E: experimental, S: Simulation](image)

6 FAILURE PREDICTION OF RECTANGULAR PLATES

In this section, experiments and numerical simulation of laterally loaded plates were conducted in order to predict the initiation and propagation of fracture. The influence of the indenter on the failure of plates was studied.

6.1 Experimental details

The experimental details are the same given in Section 5.1. However, the impact velocity is 3.8 m/s. The plate thickness is 1.4 mm (thin plate). The indenters were hemi-spherically ended projectiles of diameter 10, 16, 20 and 30 mm.

6.2 Experimental results

The force-displacement responses are shown in Figure 17. The maximum force and displacement differ for each diameter of indenter. The absorbed energy increases with the diameter of indenter. The slope of force-displacement curve decreases for the smaller indenters. The failure of the plates using smaller indenters occurred at lower displacement and force, and thus smaller portion of the incident energy was absorbed. All plates initiated fracture at peak force.

![Figure 17 Force-displacement responses.](image)

6.3 Numerical model

The numerical model is similar to the one described in Section 5.3. The failure of plate is predicted using different true stress-strain curves, element types and
boundary conditions. The specimen using an indenter of Ø 30 mm was selected to evaluate the numerical simulation.

6.4 Numerical results

First, the three true stress-strain curves (PL, GL and UN+GL) using a critical failure strain of 0.9 were used to predict the experimental response (Figure 18). All the predicted force-displacement responses were similar to the experimental response.

Models using shell and solid elements are compared in Figure 19. The material curve was defined by the UN+GL approximation. The response of shell model is better than the one of solid model, because of its higher number of integration points. The shell element has 5-integration points through the thickness and the solid element only has one.

The clamped and supported models mentioned in Section 5.3 were also used to predict the failure using a mesh size of 2.0 mm (Figure 20). It is noted that the sliding between the supports has important influence on the response of plate, decreasing the impact forces and, consequently, increasing the displacements.

7. PLASTIC RESPONSE OF STIFFENERS WITH ATTACHED PLATE

This section summarizes results from experiments and numerical simulations of stiffeners with attached plate subjected to lateral loads. The sensitivity of the incident velocity and the stiffener type is reviewed using the force-displacement response of the tested specimens.

7.1 Experimental details

The experimental tests represent a partially supported stiffener with attached plate which is struck at the mid-span by a mass. The design of the specimens (denoted by Panel A2 and Panel A3) is shown in Figure 21. The specimens were partially supported, i.e. the edges in the length direction were fully clamped whereas the edges in the width direction were free. The indenter is a hemispherically ended projectile of diameter 30 mm which uses a striking mass of 54.0 kg.
7.2 Experimental results

The resulting force-displacement responses are shown in Figure 22. It is observed that when the impact velocity increases, larger transverse displacements and impact forces are developed in both Panel A2 and Panel A3. The initial reacting forces increase with the impact velocity. The magnitudes of the maximum force, maximum deflection and permanent deflection are similar between both panels when impacted at the same velocity. The shapes of the force-displacement responses of Panel A3 show more oscillations than the ones of Panel A2, especially at the first moment of the impact.

Figure 22 Force-displacement responses

7.3 Numerical model

Four models were used (Figure 23). Two models were designed in shell and solid elements. The other two represented the weld joint using shell and solid elements in order to reproduce the experimental plastic response. The fillet weld cross-section takes the shape of a triangle and the measured leg length is 4.0 mm. The mesh size was 2.0 mm.

Figure 23 Finite element models

7.4 Numerical results

The panel A2 impacted with velocity 2.7 m/s was selected to compare the force-displacement responses (Figure 24). The maximum reacting forces are similar in the four numerical models. The ‘Shell’ and ‘Solid’ model reproduce almost the same response, overestimating the maximum and the permanent deflections. Although ‘Shell Weld’ model improves the results, the maximum and end displacements are still overestimated. The ‘Solid Weld’ model has been favorably validated against maximum force and maximum deflection. However, the permanent deflection is not accurate. Certainly, the real representation of the weld joint helps to reproduce the experimental response in this model.


8 CONCLUSIONS

This paper studied the impact strength of structural components, in order to provide the basis for the studying ship collisions. Experimental and numerical methods were used to analyze the structural components of ship subjected to lateral impact. Thus, detailed information of the impact response of structural components was obtained through drop weight impact tests and nonlinear explicit dynamic simulations.

In the analysis of plastic response of structural components, the good agreement obtained between experiments and simulations obey to a correct representation of the boundary condition. The preload force and the coefficient of friction are the most important parameters in the definition of the representation of the experimental supports.

The numerical analysis demonstrated that the results are insensitive to the mesh size and the element
type. However, the material strain rate has strong effect on the plastic response.

The indenter type has strong influence on the impact response of plate. The differences between experimental and numerical results were due to overestimation of the permanent deformation, whereas the maximum force and maximum deflection were generally very well predicted.

In the analysis of failure of structural components, the selection of mesh size and critical failure strain by numerical simulation of tensile test is valid for numerical simulation of impact test. It is a good way to predict the critical failure strain through tensile test simulation. These definitions can be applied in complex structures and to investigate the effects of all the parameters involved in this type of impact analysis.

The shape of indenter is a sensitive parameter for the failure of specimen. The small indenter is easier to break through, absorbing less energy until fracture.

REFERENCES


Parkes EW. 1958. The permanent deformation of an encastré beam struck transversely at any point in its span. Proceedings - Institution of Civil Engineers; (10): 277-304.


