Calculation of naval collisions with general use finite element software

C L Dumitrache, R Dumitrache, C Stanca, C Andrei and C Ancuta
Constanta Maritime University, Faculty of Navigation and Naval Transport,
Department of Navigation, 104 Mircea cel Batran Street, Constanta, 900663, Romania

E-mail: cosmin_dumitrache@yahoo.com

Abstract. Collisions and groundings are contributing significantly to the structural damage of ships causing almost half of the total losses of ships. The capacity of computers and software made it possible to analyse collision cases as integrated formulation where movement equations are solved for ships’ structures and for the surrounding waters by applying general numerical methods such as the finite element method. In order to ascertain the quality of software used in collision calculations, generally there are used relatively simple structures made of stiffened plates which are very similar to the structure of vessel’s side plate. In this research, for collision calculus the software ANSYS which offers two main interfaces, ADPL and WORKBENCH. It was used a procedure of explicit dynamic calculus and ANSYS software is greeting the user and requesting only the input of total time of calculus/simulation. For the explicit dynamic calculus it was used the LS-DYNA software within ANSYS, which is a very reliable and fast solution. We are using calculations and measurements of a structure from the available literature and as a conclusion, we will note that there is a good correspondence between the three categories of results: measured, calculated in the literature and the one calculated with own means.

1. Precision of collision calculation
Within this research, there were used ANSYS and LS-DYNA software. They were chosen, not because they are one of the best, but for the simple reason they are an accessible software for collision calculations.

It is most probable that finite element software like ABAQUS or DYTRAN to have much better or faster results or to have a database with more and perfected material models. The role of this paper is not to establish a ranking of the most used and best collision calculation software and unfortunately does not even exist an ideal software. These software’s are receiving constant updates or new versions not only because new options are added, but because there are corrected the errors from previous versions. A finite element software has more errors than many others because they are based on complicated mathematical models and complex numerical algorithms. A software as the ones developed by ANSYS suffers at least a major update every year and numerous minor updates.

There are numerous causes that can lead to errors in the calculation results. Below are mentioned few of the causes:

a. wrong mathematical model;
b. wrong approximation with finite element;
c. wrong implementation in the software;
d. wrong communication between the software and operating system;
e. the user filled in wrong data;
f. the user is interpreting wrong the results given by the software.

Causes a and b depend on theoretical models used, c and d depend on the way in which the software was made and only e and f depend on the knowledge and skills of the user. Quality of the software can be checked only comparing the results of the calculations with results measured on real structures tested during collisions. In this way can be checked the whole modelling process starting with mathematical model used and ending with results interpretation provided by the software.

Because of the large costs involved, there is a limited number of tests carried out on real structures. If we’ll discuss about automobiles, where series are larger and costs of the tests are relatively reduced, finite element software is used to optimise the shape of structure and in the end there are done also tests on real structures. But for the vessels, where the series are small with tremendous costs involved during tests, mostly there are used simulations with finite element to establish the collision resistance. In order to identify the quality of the software used for the collision calculations, usually there are used relatively simple structures, made of stiffened plates which are very similar to the plates of the vessel’s side plate.

For the research purpose, there were used calculations and measurements from structures used in the existing literature \[2\], \[3\] and also the results of own calculations. In the existing literature there were used modified versions of a general use program. Most of the general use programs allow the advanced users (researchers) to modify them and to add new material models. This operation is not accessible to an ordinary design engineer. Taking into account that in this paper we want to check if the activity of collision calculation can be done in similar condition as an any other finite element calculation, we preferred to use predefined material models within the software database.

### 2. Description of structure used within existing measured and calculated results

#### 2.1 Structure modelling

In the research \[2\] there are presented results of mechanical tests carried out on structures impacted on slow speed. The goal of the tests was to supply data for comparing the results with the ones got from calculation analysis and more over, to emphasize the need of stiffening the plates for an increased resistance during impact. Studied structure was made of a rectangular panel inserted in a rigid frame. Tests were carried out in three ways:

- un-stiffened panel;
- panel with one stiffener;
- panel with two stiffeners.

In figure 1 it is presented the panel with two stiffeners. In figures 2 and 3 there are presented the panel dimensions, the stiffeners and the indenter.

![Figure 1. Panel structure [2].](image-url)
2.2 Materials used in the indentation tests

In [3] are presented the results of the calculations with finite method analyse compared with the experimental results, namely bending rupture test. For yield modelling there are used two failure criteria: Rice-Tracey-Cockcroft-Latham criterion (RTCL) and Bressan-Williams-Hill criterion (BWH). The criteria are along with material routines implemented into the LS-DYNA finite element software.

For modelling with finite element there were used three meshes (5, 10, 18 mm), including Belytschko-Lin-Tsay shell elements. There should be noted the extremely fine mesh, which is accessible only if the structure has reduced dimensions, as is the present case.

It was considered a friction coefficient of 0.3 between indenter and panel, which is a reasonable friction coefficient between coated steel surfaces. Taking into account that within the tests resulted an asymmetrical fracture, it was concluded that the indenter was offset from the plate centre. Therefore, in the calculations the indenter was applied offset, at a distance of 5 mm from the centre. Also it was considered the stiffeners geometry imperfection. It was accounted a general imperfection in form of a sine curve applied to the stiffeners. The amplitude of this curve is 0.1% of stiffener length. The material is modelled using an equivalent stress-strain relationship represented by a power-law formulation which includes the plateau strain [3].
\[
\sigma_c(\varepsilon_p) = \begin{cases} 
\sigma_c, & \varepsilon_p \leq \varepsilon_{\text{plat}} \\
K(\varepsilon_p + \varepsilon_0)^n, & \varepsilon > \varepsilon_{\text{plat}}
\end{cases}
\]  \tag{1}

where \( \varepsilon_{\text{plat}} \) is the equivalent plastic strain at the plateau exit and \( \sigma_c \) is the initial yield stress. The strain \( \varepsilon_0 \) allows the plateau and power-law expression to intersect at \( (\varepsilon_{\text{plat}}, \sigma_c) \) and is obtained by:

\[
\varepsilon_0 = \left( \frac{\sigma_c}{K} \right)^{\frac{1}{n}} - \varepsilon_{\text{plat}}
\]  \tag{2}

where \( K \) and \( n \) are material parameters.

Material properties applied in the experiment are summarized in [2] where the plate parameters are: \( \sigma_c = 285 \text{MPa}, \ K = 740 \text{MPa}, \ n = 0.24, \ \varepsilon_{\text{plat}} = 0 \) and the frame parameters are: \( \sigma_c = 390 \text{MPa}, \ K = 830 \text{MPa}, \ n = 0.18, \ \varepsilon_{\text{plat}} = 0.01 \).

2.3 Comparing measured and calculated results

In this section it was observed the force-deformation relationship and the moment of initiation of fracture. In figure 4 it is presented the measured results in at the structure without stiffeners, being displayed the relation between indentation force and the indentation depth. The plate is fracturing sharply at 1500 kN and approximately 200 mm. In figure 5 it is presented overlaid the measured results during the tests [2] and calculated with finite element method [3].

![Figure 4. Relation between force indentation and indentation depth [2].](image1)

![Figure 5. Measured and calculated results [3].](image2)

3. Own calculations with a finite element software

3.1 General information about calculation

Calculations were carried out with ANSYS LS-DYNA using Workbench interface. ANSYS Workbench LS-DYNA combines the power of the LS-DYNA solver with pre-processing and post-processing tools available in the ANSYS mechanical environment.

To define geometry, it was used an object “Geometry” type, an object “Explicit Dynamics” for calculation with ANSYS software and an object “Explicit Dynamics (LS-DYNA) export” for
calculation with LS-DYNA software. For an easier input of data on changing of mesh there were used copies of the object “Explicit Dynamics”, which were modified as necessary.

![Structure Geometry](image)

**Figure 6.** The structure geometry.

Description of the structure, from mechanical point of view (loads, nodes) was carried out also in Workbench.

For the plate, for the frame and for the indenter (considered rigid in the calculation) there were used the following materials:

![Steel Plate Material](image)

**Figure 7.** Description of the steel plate material.
Figure 8. Description of the steel frame material.

Figure 9. Description of the indenter material.

Depending on the kind of calculation, there were used also other values of the “Maximum Equivalent Plastic Strain EPS” of the steel plate. Of the indenters’ properties, the density is one of the most important and it was chosen in such a way to allow a greater time step.

3.2 Calculations on various mesh sizes
There were made calculations for 50 mm, 20 mm and 10 mm mesh. Further, it will be presented the calculations and results for 20 mm and 10 mm mesh.

The structure is made of 15,792 “shell” type elements (Belytschko-Tsay) and 15,727 nodes. The time step size was 0.02 s and the speed of indenter of 10 m/s, parameters which leads to a maximum indentation of 0.2 m. The problem was solved using 3 CPU’s in 2 hours 14 minutes. Initial time step of the simulation was $\Delta t = 1.7 \cdot 10^{-3} ms$ and for integration from 0 to 20 ms were necessary 117,732 steps. Time step size can be estimated with:
It is observed that the time step used by the software is about 20 times smaller than the estimated one and this thing leads to an estimated calculated time 20 times bigger. Taking into account that the indenter in somehow rigid and is moving with constant speed, it can be increased the density of the material of the indenter in such a way to increase the time step size. In accordance with the formula to increase the time step 20 times, the ratio $\rho / E$ should increase 400 times. Because the indenter is considered rigid, the Young modulus $E$ can be changed as well. After changing the density, it was made again the calculation and it was observed that the shortest time step it is calculated on the central plate elements. Calculation time was reduced, from more than 2 hours initially, to under six minutes.

Maximum equivalent plastic strain for 20 mm mesh it is being calculated with the formula:

$$\varepsilon_f = 0.08 + 0.65 \frac{f}{l}$$

and has the value $\varepsilon_f = 0.2425$.

Time step calculated by the program from the smallest element from the structure was: $\Delta t = 2.162 \times 10^{-6} s$. 

$$\Delta t \approx \frac{\rho}{E} \sqrt{\frac{7896}{2.1 \times 10^{11}}} s = 3.49 \times 10^{-6} s = 3.49 \times 10^{-3} ms$$

(3)
Time step size estimated with formula (3) was \( \Delta t = 2.47 \times 10^{-6} \) s and the one indeed used by the software was \( \Delta t = 1.079 \times 10^{-6} \) s.

Maximum equivalent plastic strain for 10 mm mesh was calculated with the formula (3) and has the value \( \varepsilon_f = 0.405 \).

**Figure 12.** Equivalent plastic strain at 10 mm mesh with ANSYS.

**Figure 13.** Equivalent stress at 10 mm mesh with ANSYS.

**Figure 14.** Contact force variation during 20 mm mesh calculation with ANSYS.
Figure 15. Contact force variation during 10 mm mesh calculation with ANSYS.

For the calculation with LS-DYNA, after exporting the data from Workbench, it was used the software “ANSYS Mechanical APDL Product Launcher” to specify how the calculation was done. In this software, apart the information about licence, working and input folders, there are information about how the calculation in distributed on more processors, even more computers.

Having available the simplest method, where only one computer is used, there area two possibilities:

a) we are using the option from the page “Customization/Preferences” to pinpoint the number of processors (cores) which will be used in the calculation (“Number of CPU’s”)

b) we are using the options from the page “High Performance Computing Setup” where are activated the options “Use Distributed Computing (MPP)” and “Use Local Machine Only” and is specified “Number of Processors”.

Of the two above mentioned possibilities, the first should be faster. Unfortunately, sometimes first procedure is not working, therefore it is recommended for both of them to be tested and to be chosen the most optimal one.

The file exported by Workbench must be modified to show the contact forces calculation.

For the 20 mm mesh calculation, the time step size used by LS-DYNA was 3.0083E-03 ms. The calculation was carried out using option b) above mentioned and the total elapsed run time is shown as below, extracted from the output file of the software.

For the 10 mm mesh calculation with LS-DYNA, it was used a PC with i5 processor and four cores. There were simulated four computers on the same PC, one for every each core of the processor. Total elapsed run time dropped approximately four times as if it was used a single CPU. It is worth to compare also the total run time with ANSYS of about 71 minutes with the total time with LS-DYNA, which is under 16 minutes.

“Problem time $= 2.5001E+01$
Problem cycle $= 8581$
Total CPU time $= 113$ seconds ( 0 hours 1 minutes 53 seconds)
CPU time per zone cycle $= 758$ nanoseconds
Clocktime per zone cycle $= 762$ nanoseconds
Parallel execution with 4 MPP proc”
Figure 16. Equivalent plastic strain at 20 mm mesh with LS-DYNA.

Figure 17. Equivalent stress at 20 mm mesh with LS-DYNA.

The software calculated and used the time step size $\Delta t = 1.5041E-03 \text{ ms}$.

The total elapsed run time is shown as below, extracted from the output file of the software:

4. Conclusions

Based on existing references, there were presented results of the measurements of a structure which is subject to an impact. There were also presented calculated results using references [2], [3]. The results of the own calculations are presented in table 1, below. It was considered that the most important results are the maximum contact force and the indentation caused.

Differences between results are coming mainly from the way how local yielding of the material is appreciated. In references [2] and [3] there are used algorithms based on the idea of erosion of the material. These algorithms were implemented in LS-DYNA by the respective authors, having a research purpose.

In own calculations, local fracture of the material is determined based on the equivalent plastic strain and it is considered that the material fractures if the strain is reaching a certain limit. It was chosen this type of appreciation because the reason was to use the software ANSYS and LS-DYNA in their standard form, taking into account that modifying of a complex software as LS-DYNA is not accessible to any ordinary user.

We may say, without making a wrong supposition, that the yielding equivalent plastic strain is not a material dependent characteristic, but a mesh one.

"Problem time = 2.0001E+01
Problem cycle = 13341
Total CPU time = 929 seconds (0 hours 15 minutes 29 seconds)
CPU time per zone cycle = 1051 nanoseconds
Clocktime per zone cycle = 1052 nanoseconds
Parallel execution with 4 MPP proc"
Table 1. Differences in percentage between own results and references results.

|                | Measured values [2] | Difference to measured values [%] | Calculated values [3] | Difference to calculated values [%] |
|----------------|----------------------|-----------------------------------|------------------------|-------------------------------------|
|                | F [kN]    | Indent [mm] | F [kN]    | Indent [mm] | F [kN]    | Indent [mm] | F [kN]    | Indent [mm] | F [kN]    | Indent [mm] |
| Own results calculated with ANSYS | 50 mm mesh | 730      | 110      | 50.68      | 45.00      | 37.06      | 35.29      |
|                | 20 mm mesh | 1123     | 156      | 24.12      | 22.00      | 3.19       | 8.23       |
|                | 10 mm mesh | 1440     | 190      | 2.70       | 5.00       | 24.13      | 11.76      |
| Own results calculated with LS-DYNA | 20 mm mesh | 670      | 110      | 54.73      | 45.00      | 42.24      | 35.29      |
|                | 10 mm mesh | 1045     | 160      | 29.39      | 20.00      | 9.91       | 5.88       |

5. References

[1] Dumitrache C L 2015 Research on the Ship’s Collision Mechanics and Resulting Damages (PhD Thesis) (Constanta: Constanta Maritime University)

[2] Alsos H S and Amdahl J 2009 On the resistance to penetration of stiffened plates, Part I: Experiments Int. J. Imp. Eng. 36 799–807

[3] Alsos H S, Amdahl J and Hopperstad O S 2009 On the resistance to penetration of stiffened plates, Part II: Numerical analysis Int. J. Imp. Eng. 36 875–87
[4] Gavrilescu I and Ionas O 1997 Numerical definition of the hull shape *The Annals of “Dunarea de Jos” University of Galati Shipbuilding* XI

[5] Gavrilescu I 1997 Modeling of naval structures by plate in elasto-plastic domain *The Annals of “Dunarea de Jos” University of Galati Applied Mechanics* X

[6] Servis D, Samuelides M, Louka T and Voudouris G 2002 Implementation of finite elements codes for the simulation of ship-ship collisions *J. of Ship Research* 46(4) 239-47

[7] Domnişoru L, Gâvan E and Popovici O 2005 *Analiza Structurilor Navale prin Metoda Elementului Finit* (Bucuresti: Ed Didactica si Pedagogica) p 216

[8] Kitamura O 1997 Comparative study on collision resistance of side structure *Marine Technology* 34(4) 293-308