Materials used in a naval collision calculation with finite element software

C L Dumitrache, C Faitar, N Buzbuchi and R Dumitrache

1 Constanta Maritime University, Faculty of Navigation and Naval Transport, Department of Navigation, 104 Mirea cel Batran Street, Constanta, 900663, Romania
2 Constanta Maritime University, Faculty of Naval Electromechanics, 104 Mirea cel Batran Street, Constanta, 900663, Romania

E-mail: cosmin_dumitrache@yahoo.com

Abstract. Collisions and groundings have a significant contribution to structural damages occurring to ships. For collision calculation are used non-linear relations between strains and deformations, considering a significant difference between the initial shape and final deformed shape of the structure, and the mechanical equilibrium equations are written on the deformed shape of the structure. Using a mechanical test to a material sample will result the connection between the engineering strain and engineering deformation. During numerical simulation, we are using the real mechanical characteristic curve, which represents the reliance between real strains and real deformations, those being calculated from the distorted material sample. For calculation, we are using Johnson – Cook plasticity model, frequently used by other authors as well during collision calculation. There is included new information, also tabular calculations using specific formulas, resulted during carried out studies, to establish the model parameters considering the measured data or using information from another material model (exponential law). Considering the materials used within ANSYS software and calculus processor LS-DYNA, are presented types of materials useful for collision calculation, describing the elasticity, plasticity and yielding. Following, it is used a procedure for dynamic explicit calculus because it is suitable for solving of nonlinear systems. During simulation with finite element of the collision, there are no strains and deformations concentrations and it is necessary to generate an artificial curve, which leads to same results, as in reality and as well finite element analysis.

1. Introduction
For the numerical simulation of ships’ collision, it is usually used the finite element method. In order to employ this method, it is necessary to have, among other things, information about the behaviour of used materials that is as realistic as possible. During the collision process the material is subject to large deformations and possibly fractures and, therefore, the description of the materials could be rather complex. The behaviour of a material is described using the strain-deformation curve (characteristic curve). A first shape of this curve is obtained by measuring. From the mechanical testing on a material sample will result the connection between the engineering strain and the engineering deformation (conventional characteristic curve). Engineering strains and deformations are calculated on the initial shape of the material sample and therefore the conventional curve is not used in simulation.
Numerical simulation uses the real mechanical characteristic curve which represents the dependence between real strains and real deformations. The real curve is deducted from the conventional curve by using simple conversion relations. The real strains and deformations are calculated on the distorted material sample. The real curve thus determined is correct until necking occurs (until the ultimate test strength or the ultimate engineering tension). In order to obtain the real characteristic curve for deformations larger than the necking deformation, additional calculations are necessary, such as simulations with the finite element method of the testing process. In order to have a complete description of the material it is necessary to define the fracture deformation or an algorithm for the progressive reduction of the material rigidity, which leads to fracture.

Numerous causes can lead to errors in the calculation results. Following are mentioned some 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 will 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 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. Material failure modelling

The behaviour of ship’s structure during collision depends, to the largest extent, on the fracture of the hull and frame. These fractures take over the largest part of the energies that arise in the collision process [1]. As such, when performing simulations through calculi, it is important to use correct information about the materials’ behaviour in the non-linear calculus and, especially, realistic failure criteria. When analysing the ship’s strength to collision the most important parameter is the critical energy, namely the energy necessary to break the shell or the bottom or to break the double board or the double bottom, as the case may be. This critical energy is essentially dependent on the fracture deformation (yielding) [1]. In [1] it is shown that the doubling of the fracture deformation leads to (approximately) the doubling of the critical energy. The fracture of a structural element is a very complicated process which is influenced by material’s features, strain, building process, environmental conditions, operation mode, etc. In the case of a finite element analysis, the fracture depends additionally on the fineness of the discretization, the elements’ shape and type, etc.
2.1. Constant yielding deformation

The Germanischer Lloyd engineering guidelines [2] recommend the use of the constant yielding deformation (the actual plastic deformation) in the collision calculus, with the value:

\[ \varepsilon_f = 0.05 \]  

(1)

This low value (1) implies the acceptance of some reduced plastic deformations before the fracture. In correspondence to this deformation, the structure subject to collision can absorb only a small quantity of energy. A major disadvantage of this rule resides in the fact that it does not take into account the structure’s finite element discretization. The calculus has shown in practice that there is a strong connection between the structure’s discretization and the yielding deformation.

2.2. Yielding deformation dependent on discretization

Calculus simulation is performed using the finite element method and the fineness of discretization influences the results (deformations and strains). When the fineness of discretization is increased, so are the deformations in the most stressed areas of the structure (in the “stress concentrators”). This is a normal phenomenon, where the increase of network density leads to results that are closer to reality. The increase of network density also causes the increase of the calculation effort, which means that it would be preferable to obtain realistic results (as confirmed by tests) by using finite element networks as rough as possible. According to an empiric rule known in this field, by reducing the elements’ dimension by half, one can increase the computation time by 8 times. Typical discretization used to simulate ship collisions employs elements having a side comprised between 0.1 m and 1 m [3]. For the sake of comparison, when simulating automobile collisions, are used finite elements with the side comprised between 6 and 10 mm, given their fairly reduced dimensions. Typically, the ratio between the elements’ side length and their thickness is comprised between 10 and 60. In conclusion, it is preferable to use a limit deformation which depends on the finite elements discretization. Sequel to numerous tests, it is proposed the relation for the yielding deformation computation with shell type elements [4]:

\[ \varepsilon_f = 0.08 + 0.65 \frac{t}{l} \]  

(2)

where \( \varepsilon_f \) is the yielding deformation, \( t \) is the thickness of the shell, and \( l \) is the elements’ dimension.

Although still not perfect, the relation expressed in formula (2), reflected in figure 1 is extremely useful, especially if the program used to compute does not provide other means to treat material yielding.

**Figure 1.** Dependence between discretization and yielding deformation. [4]

**Figure 2.** Relation proposed by Kitamura for yielding deformation. [3]
It is proposed a similar relation useful when employing calculus with beam or truss elements [4]:

\[ \varepsilon_f = 0.079 + 0.76 \frac{f}{l} \]  

To the same purpose, Kitamura [3] proposes a rather similar way to calculate the shell yielding deformation, represented in figure 2, which actually shows the evolution of yielding deformation based on elements’ length.

2.3. Johnson – Cook plasticity model
2.3.1. Strain model. In the Johnson-Cook plasticity model the yielding point is expressed by the relation:

\[ \sigma_t(\tilde{\varepsilon}^p, \tilde{\varepsilon}^f, T) = [A + B(\tilde{\varepsilon}^p)^n][1 + C \ln \dot{\varepsilon}][1 - (T^*)^m] \]  

where \( A, B, C, n, m \) are constants which depend upon the material and can be determined by measurements. The first parenthesis in relation (4) refers to quasi-static behaviour, the second contains the influence of the deformation speed, and the third to the influence of temperature. The following denotations are used:

\( \tilde{\varepsilon}^p, \tilde{\varepsilon}^f \) equivalent plastic strain and equivalent plastic strain rate;
\( \dot{\varepsilon}^* = \frac{\tilde{\varepsilon}^f}{\dot{\varepsilon}_0} \), dimensionless equivalent strain rate; \( \dot{\varepsilon}_0 \) is the reference strain rate; Johnson and Cook had measurements for \( \dot{\varepsilon}_0 = 1 \text{s}^{-1} \).

\( T^* \) is the dimensionless equivalent temperature, defined by:

\[ T^* = \frac{T - T_{\text{room}}}{T_{\text{melt}} - T_{\text{room}}} \]  

where: \( T \) temperature [K], \( T_{\text{room}} \) reference ambient temperature [K], \( T_{\text{melt}} \) melting temperature [K].

It should be noted that \( A \) is the initial yield stress [Mpa].

A sample is described by eight material parameters: \( A, B, C, n, m, \dot{\varepsilon}_0, T_{\text{melt}}, T_{\text{room}} \).

The relation for computing the yield stress was proposed by Johnson and Cook [5], and their paper presents constant material examples, as well as the description of practical procedures used in order to establish the material constants.

2.3.2. Failure model. The Johnson-Cook elastic-plastic strain model is associated to the Johnson-Cook failure model, in which failure occurs after the accumulation of damages. The current damage coefficient is computed using the relation:

\[ D = \sum \frac{\Delta \tilde{\varepsilon}^p}{\varepsilon^f} \]  

where \( \Delta \tilde{\varepsilon}^p \) are the equivalent plastic strain increments, corresponding to the integration steps, and \( \varepsilon^f \) are the equivalent fracture strains. It is thought that the fracture occurs when the damage parameter \( D \) reaches value 1. Additionally, it may be observed that the Johnson-Cook model presented herewith considers that the material is failing when damage begins (when \( D = 1 \)).

The equivalent fracture strain is computed with:

\[ \varepsilon^f = [D_1 + D_2 \dot{\sigma} \sigma^*][1 + D_4 \ln \dot{\varepsilon}^*][1 + D_5 T^*] \]  

where \( D_i, i = 1, 2, ..., 5 \) are material constants and \( \sigma^* \) is the ratio between pressure and actual stress (the stress triaxiality factor):

\[ \sigma^* = \frac{p}{\sigma} \]
A material is described by eight material parameters: \( D_1, D_2, D_3, D_4, D_5, \dot{\varepsilon}_0, T_{\text{melt}}, T_{\text{room}} \).

The simplest version of this fracture model is characterized by:

\[
D_2 = D_3 = D_4 = D_5 = 0
\]  

Namely, the equivalent fracture strain is computed with:

\[
\varepsilon' = D_1
\]

**Table 1.** Examples of steels associated to the Johnson-Cook model. [5]

| Material          | Material parameters |
|-------------------|--------------------|
| Weldox 460 E STEEL | \( D_1 = 0.0705 \) \( D_2 = 1.732 \) \( D_3 = -0.54 \) \( D_4 = -0.015 \) \( D_5 = 0 \) \( \dot{\varepsilon}_0 = 5 \times 10^{-4} \) \( T_{\text{melt}} = 1800 \) \( T_{\text{room}} = 293 \) |
| 4340 STEEL        | \( D_1 = 0.05 \) \( D_2 = 3.44 \) \( D_3 = -2.12 \) \( D_4 = 0.002 \) \( D_5 = 0.61 \) \( \dot{\varepsilon}_0 = 1 \) \( T_{\text{melt}} = 1793 \) |

The Johnson-Cook material model, detailed in table 1 can be applied to most metals and is valid with both high and low strain rates and even in the quasi-static case. Typical applications are: study of explosion effects, ballistic penetration, impact.

2.3.3. Algorithm: Simplified computation of Johnson-Cook model parameters. As an example, in table 2 and table 3 it is presented the computation of Johnson-Cook model parameters used in the collision calculations presented in this paper.

**Table 2.** Computed parameters of simplified Johnson-Cook material model. [6]

| Data | Real stresses and strains | Model parameters |
|------|---------------------------|------------------|
| \( E = 2.10 \times 10^{11} \) Pa | \( \sigma_u = 1.08 \times 10^{09} \) Pa | \( A = 3.81 \times 10^{08} \) Pa |
| \( \sigma_{0c} = 3.80 \times 10^{08} \) Pa | \( \varepsilon_u = 0.3001046 \) | \( n = 4.56 \times 10^{-01} \) |
| \( \sigma_{0u} = 8.00 \times 10^{08} \) Pa | \( \sigma_{0u} = 1.81 \times 10^{-03} \) | \( B = 1.22 \times 10^{09} \) Pa |
| \( \varepsilon_{0w} = 3.50 \times 10^{-01} \) | \( \sigma_{c} = 3.81 \times 10^{08} \) Pa | \( \varepsilon_{pw} = 2.95 \times 10^{-01} \) |

**Table 3.** Computation of simplified Johnson-Cook model parameters. [6]

| Excel rows | Excel columns |
|------------|---------------|
| A          | B             | C |
| 1 Data     |               |   |
| 2 E        | 2100000000000 | Pa |
| 3 \( \varepsilon_{0c} \) | 380000000 | Pa |
| 4 \( \varepsilon_{0u} \) | 800000000 | Pa |
| 5 \( \varepsilon_{0w} \) | 0.35 |   |
| 6 Real stresses and strains |               |   |
| 7 \( \sigma_u \) | =B4*(1+B5) | Pa |
| 8 \( \varepsilon_u \) | =LN(1+B5) |   |
| 9 \( \varepsilon_{0c} \) | =B3/B2 |   |
3. Estimation of Johnson – Cook model parameters

To model the materials used in the computations presented by Hagbart S. Alsos et al [7, 8], a modified exponential law was used. In this paper ANSYS and LS-DYNA computation programs are used. None of these programs has the respective materials implemented by default. In principle, LS-DYNA allows the addition of material models and this is what the authors of the article have done [2, 3]. This implies FORTRAN coding and program recompilation naturally followed by testing. Such approach is typical of research activity. Here, we would like to find out if the collision calculus can be carried out with the available instruments, without altering them. Namely, we would like to establish if engineers who only have basic skills of using finite element software could perform a collision calculus. As such, instead of adding a new material model to the LS-DYNA material library, we will prefer to use an already existing material.

3.1 Estimation of model parameters starting from an exponential law

The Johnson-Cook model will be used from the LS-DYNA material models library and, therefore, its parameters will be estimated starting from the exponential law parameters used by Hagbart [7, 8]. The issue can be approached in several ways, but the easiest way seems to be the use of the method of nonlinear least squares. As such, we will presume we know the exponential law parameters [7, 8]:

|   | $\sigma_c$ | $=B3*(1+B9)$ Pa |
|---|------------|-----------------|
| 10| $\varepsilon_{pu}$ | $=B8-B7/B2$ |
|   | **Model parameters** | |
| 11| A | $=B10$ Pa |
| 14| n | $=B7*B11/(B7-B10)$ |
| 15| B | $(B7-B10)/B11^nB14$ Pa |

Figure 3 represents the corresponding curve $\sigma_c(\varepsilon_p)$.

![Johnson-Cook hardening curve](image)

**Figure 3.** The Johnson-Cook hardening curve. [6]
Using this relation, we can compute the levels:
\[ \sigma_{\text{exp}}(\varepsilon_p) = \begin{cases} \sigma_c, & \varepsilon_p \leq \varepsilon_{\text{plat}} \\ K \left( \varepsilon_p + \varepsilon_0 \right)^n, & \varepsilon > \varepsilon_{\text{plat}} \end{cases} \] (11)

For some arbitrary values \( \varepsilon_{pl} \) of the real plastic strain, Johnson-Cook model parameters are \( A, B, m \). To avoid confusions parameter \( n \) of the Johnson-Cook model was noted here \( m \) since the exponential law also contains a parameter \( n \). Therefore, the Johnson-Cook hardening curve is expressed using relation (13):
\[ \sigma_{\varepsilon,JC}(\varepsilon_p) = A + B \left( \varepsilon_p \right)^m \] (13)

Parameter \( A \) has the value:
\[ A = \sigma_c \] (14)

where \( \sigma_c \) is a stress yield (initial).

For parameters \( B, m \) some arbitrary values are imposed initially (at start). The correct (real) values of parameters \( B, m \) will be determined iteratively through the nonlinear least squares method.

With the initial values \( B, m \) or those from a certain step of the least squares algorithm the stress values are computed in accordance with the Johnson-Cook model:
\[ \sigma_{\varepsilon,JC}(\varepsilon_p) = \sigma_{\varepsilon,JC}(\varepsilon_p), i = 1, 2, \ldots \] (15)

Within the least squares algorithm parameters \( B, m \) are determined in such a way that between the values \( \sigma_{\varepsilon,JC} \) and \( \sigma_{\varepsilon,JC} \) the differences are as small as possible. To this purpose, the sum is made:
\[ S = \sum_i \left( \sigma_{\varepsilon,JC} - \sigma_{\varepsilon,JC} \right)^2 \] (16)

and the optimization issue is solved without limiting conditions:
\[ \min_{B, m} S \] (17)

It is fairly easy to recognize the problem of the nonlinear least squares. The problem is easily solved, for example in Excel, by using the component „Solver add-in”.

Calculus setting in Excel is expressed as in table 4:

**Table 4. Parameters of material model computed using an exponential law.** [6]

| Excel computation of parameters |
|-------------------------------|
| Sc 285                        |
| B 497.0489942                 |
| K 740                         |
| m 0.489106223                 |
| n 0.24                        |

| eps | L E  | J C  | err %  | (J C - L E)^2 |
|-----|------|------|--------|---------------|
| 0   | 285  | 285  | 0      | 0             |
| 0.1 | 443.770523 | 446.1733 | 0.53852 | 5.773118222 |
| 0.2 | 513.839179  | 511.2188  | -0.51258 | 6.866586742 |
| 0.3 | 562.425793  | 560.8392  | -0.2829  | 2.517344452 |
| 0.4 | 600.488971  | 602.515   | 0.336264 | 4.104847192 |

\[ S = 19.26189661 \]
Bold and italic characters are used to highlight the optimization problem variables $B, m$ and the objective function $S$ (table 5).

Table 5. Computation of material model parameters using an exponential law. [6]

| Excel rows | Excel columns |
|------------|---------------|
| A | B | C | D | E |
| 1 | Sc | 285 | | $B$ | 497.048994214211 |
| 2 | K | 740 | | $m$ | 0.4891062227268 |
| 3 | n | 0.24 | | | 75 |

The graphics of the two curves is expresses in figure 4:

![Exponential model and Johnson-Cook model](image)

Figure 4. Exponential model and Johnson-Cook model. [6]

To sum up, the hardening curve expressed by relation (13) is characterized by material constant $A$ defined by (14) and by constants $B, m$ with the values (18):

$B = 497 MPa$

$m = 0.389$

To shed more light to the issue, it is then estimated the stress and plastic deformation corresponding to the beginning of necking, also known as ultimate plastic stress and strain (or fracture stress and strain in strength of materials course books). Necking is characterized by the condition (of maximum):
\[
\sigma_e(\varepsilon_{pu}) = -\frac{\partial \sigma_e(\varepsilon_{pu})}{\partial \varepsilon_{pu}} \tag{19}
\]

where:

\[
\frac{\partial \sigma_e(\varepsilon_p)}{\partial \varepsilon_p} = mB(\varepsilon_p)^{m-1} \tag{20}
\]

and \(\varepsilon_{pu}\) is the plastic strain at the beginning of the necking process.

The equation (19) is nonlinear. The equation is easily solved, by using again the „Solver” add-in in the EXCEL program. The EXCEL Solver solves optimization problems and, therefore, the equation (19) becomes an optimization problem without limiting conditions:

\[
\min_{\varepsilon_p} \left( \sigma_e(\varepsilon_p) - \frac{\partial \sigma_e(\varepsilon_p)}{\partial \varepsilon_p} \right)^2 \tag{21}
\]

As solution of the optimization problem as in relation (21), the ultimate plastic strain \(\varepsilon_{pu}\) and the ultimate stress \(\sigma_u = \sigma_{uy}\) are obtained. Based on these, we can determine:
- the true ultimate strain \(\varepsilon_u\):
  \[
  \varepsilon_u = \varepsilon_{pu} + \frac{\sigma_u}{E} \tag{22}
  \]
- engineering ultimate strain \(\varepsilon_{0u}\):
  \[
  \varepsilon_{0u} = e^{\varepsilon_u} - 1 \tag{23}
  \]
- engineering ultimate stress \(\sigma_{0u}\):
  \[
  \sigma_{0u} = \frac{\sigma_u}{1 + \varepsilon_{0u}} \tag{24}
  \]

Next, the EXCEL sheets presented in tables 6 and 7, are used to determine the stress \(\sigma_{0u}\) and the engineering ultimate strain \(\varepsilon_{0u}\):

**Table 6. Stress \(\sigma_{0u}\) and engineering ultimate strain \(\varepsilon_{0u}\).** [6]

| Excel computation of parameters |  |
|---------------------------------|--|
| E                              | 2.10E+11 MPa |
| A                              | 285 Mpa |
| B                              | 497 MPa |
| m                              | 0.489 |

**Epspu** 0.222775231

| Sc                              | 523.4862278 |
| dSc/dEpsp                      | 523.4862278 |
| (Sc-dSc/dEpsp)^2              | 4.04003E-20 |

| Epsu                           | 0.222775234 |
| Eps0u                          | 0.249539688 |
| S0u                            | 418.943258 |
Table 7. Determining the stress $\sigma_{uu}$ and the engineering ultimate strain $\varepsilon_{0u}$ [6]

| Excel rows | Excel columns |
|------------|---------------|
| A          | B             |
| 1          | E             |
| 2          | A             |
| 3          | B             |
| 4          | m             |
| 5          |               |
| 6          | $\varepsilon_{pu}$ | $0.222775231128317$ |
| 7          |                |
| 8          | Sc             |
| 9          | $\frac{dSc}{dEpsp}$ | $mB6^m$ |
| 10         | $(Sc-dSc/dEpsp)^2$ | $(B8-B9)^2$ |
| 11         |                |
| 12         | $\varepsilon_{pu}$ | $B6+B8/E$ |
| 13         | $\varepsilon_{0u}$ | $\exp(B13)-1$ |
| 14         | $S_{0u}$       | $B8/(1+B14)$ |

3.2 Material models implemented in ANSYS and LS-DYNA

In ANSYS a large number of material models are predefined and the software also contains a library of material constants. The selection of material models has also taken into account the fact that the collision calculus generally uses shell type elements and, consequently, the number of models offered by ANSYS is reduced. For example, to model a fracture, it can only be used the real fracture plastic strain with its alternatives, as presented further in this chapter. LS-DYNA has almost 200 predefined material models but, if a selection is made for their use in the collision calculus for ships modelled with shell type elements, the number of these models is greatly reduced.

ANSYS models are essentially enough for an engineering collision modelling. Surely, both programs allow the description of new material models.

3.2.1. Elasticity. To describe elastic behaviour in ANSYS, the option „Linear elastic” is used, within which “Isotropic Elasticity” is selected, if the material has an isotropic behaviour. This is the standard option when working with metallic structures. Isotropic linear elastic behaviour is described by the longitudinal elasticity model $E$ and the Poisson coefficient $\nu$. Based on this data the software computes the bulk modulus and the shear modulus. Alternatively, it can be used the orthotropic material model, described by three each longitudinal elasticity moduli, shear moduli and contraction coefficients, nine material constants in total.

3.2.2. Plasticity. In order to describe the plastic behaviour in ANSYS, several material versions can be used.

1. Bilinear material model with isotropic hardening. Parameters $A, \sigma_y$ are introduced, as defined by the simplified hardening curve:

$$\sigma = \sigma_y + A\varepsilon_p$$

where $\sigma_y$ is the yield stress, $\varepsilon_p$ is the plastic strain, $A$ is the plastic modulus (also called tangent modulus). It should be noted that the kinematic hardening model couldn’t be used with shell elements.
The difference between the two models, kinematic and isotropic, is given by the current interval of elastic stress. The difference between the kinematic and isotropic models becomes visible only when, during the stress, some discharge occurs. The kinematic hardening model is useful to model the Bauschinger effect and is successfully employed for shell modeling. In the case of isotropic hardening, the current elastic stresses are comprised in the range \([-\sigma, \sigma]\), where \(\sigma\) is the current stress corresponding to the plastic strain. In the case of kinematic hardening elastic stresses are comprised in the range \(\sigma - 2\varepsilon_p\). To describe such a material in ANSYS, “Plasticity” option is used, within which either “Bilinear isotropic hardening” or “Bilinear kinematic hardening” are selected.

2. Multilinear model with isotropic hardening. This model is a generalized bilinear model. It is defined by the hardening curve \(\sigma(\varepsilon_p)\) as a linear portion function. To describe such a material in ANSYS, “Plasticity” option is used, within which “Multilinear isotropic hardening” is selected. ANSYS also offers the “Multilinear kinematic hardening” option, but it is not implemented for shell elements.

3. Johnson – Cook model. Material constants which describe the Johnson-Cook hardening curve are introduced. The “Plasticity” option is used, with the “Johnson-Cook strength” option. The hardening curve \(\sigma(\varepsilon_p)\) is defined as a function of plastic strains, deformation rate and temperature.

4. “Cowper Symonds Strength” model, which is similar to the Johnson-Cook model, but the hardening curve is defined as a function of only plastic strains and deformation rates. So, in this case, temperature is not involved.

5. Other material models, such as Zerilli-Armstrong and Steinberg-Guinan are also implemented in ANSYS.

4. Conclusions
Given that in the specific literature, information about various materials (material constants) is sparse, in this paper we suggest a way to determine material characteristics. The original presentation includes a series of Excel sheets that may be used to estimate the Johnson-Cook material model parameters, which is the material model frequently employed in the collision calculations. During the finite element collision simulation stress and strain concentrations does not occur and therefore an artificial characteristic curve should be generated, which should lead to the same results (forces and moments) both in reality and finite element analysis. This artificial curve has a fracture strain markedly different from the real fracture strain. For this reason, the equivalent plastic strain used in the finite element calculation to simulate the material is a mere characteristic of finite element discretization, rather than a material characteristic.

For example, in a Ship Structure Committee report from 2002, related to modelling structural damage in ship collision, it is observed that, out of more than 100 material models implemented in LS-DYNA, only three are useful for the collision calculation: the elastic-plastic model with isotropic or kinematic hardening, the multi-linear elastic-plastic model with isotropic hardening and the rigid material model. Material models offered by ANSYS are also offered by LS-DYNA, but not vice-versa.

In general, to model failure (fracture) in ANSYS, two aspects must be taken into account:
- failure initiation; when a criterion for failure initiation is being fulfilled, a procedure of post-failure computation is activated;
- behaviour modelling after failure initiation; after failure initiation it is thought that an immediate fracture will occur or that a gradual damage will take place until the final fracture. In either case failure (the fracture) has a local feature within the respective finite element. When using shell elements, ANSYS software doesn’t provide too many options. To characterize the fracture, the plastic failure strain can be used.

It is considered that failure occurs as soon as the actual (=equivalent) plastic strain exceeds the provided maximum value. To use it, within the “Failure” option “Plastic Strain Failure” is selected.
Alternatively, similar options may be used: “Principal Strain” or “Shear Strain”, namely the principal deformation or the shear deformation.

If choosing ANSYS software was rather subjective and facilitated by having access to a program able to perform collision calculi, choosing the LS-DYNA program was inevitable since this is the standard program used to perform dynamic explicit calculi. Although the presentation has a practical feature, detailed information is given only in relation to materials’ study. As far as the method of operation is concerned, it is recommended the use of ANSYS WORKBENCH for data preparation, LS-DYNA for computation and LS-PREPOST for result analysis.

5. References

[1] Peschmann J 2001 Berechnung der Energieabsorption der Stahlstruktur von Schiffen bei Kollision und Grundberührung, PhD thesis, Institute for Ship Structural Design and Analysis, Hamburg University of Technology

[2] Germanischer Lloyd 2010 Strengthening against Collisions, Part 1 - Seagoing Ships, Chapter 1 - Hull Structures, Section 33

[3] Kitamura O 2001 FEM Approach to the Simulation of Collision and Grounding Damage, 2nd International Conference on Collision and Grounding of Ships, Copenhagen, Denmark

[4] Scharrer, Zhang, Egge 2002 Kollisionsberechnungen in schiffbaulichen Entwurfssystemen, Abschlußbericht zum Vorhaben MTK0614, Bericht Nr. ESS 2002.183, Germanischer Lloyd

[5] Johnson G R, Cook W H 1983 A constitutive model and data for metals subjected to large strains, high strain rates and high temperatures, Proceedings of the 7th International Symposium on Ballistics 541-547

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

[7] 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

[8] 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

Acknowledgement

„This work is supported by the project ANTREPRENORDOC, in the framework of Human Resources Development Operational Programme 2014-2020, financed from the European Social Fund under the contract number 36355/23.05.2019 HRD OP /380/6/13 – SMIS Code: 123847.”