Aspects regarding the increase of the protection level of the falling-object protective structures (FOPS) by using the post-critical behavior of the support elements

P Dumitrache\textsuperscript{1,2,*} and A M Goanță\textsuperscript{1,2}

\textsuperscript{1} “Dunarea de Jos” University of Galati, Engineering and Agronomy Faculty of Braila, Calea Calarasilor Street, 810017, Braila, Romania

\textsuperscript{2} Research Centre for Mechanics of the Machines and Technological Equipments, Calea Calarasilor Street, 810017, Braila, Romania

Email: dumitrache.petru@gmail.com; pdumitrache@ugal.ro

Abstract. For earth-moving machinery which are operated in areas with danger of falling objects on its cabin, an important performance requirements imposed on the machine is the protection of the human operator in case the cabin is hit by a falling object. As a result, the mechanical system formed by the cab's strength structure and its supporting elements must have the ability to dissipate high impact energies, so that the deformed system does not penetrate the deflection limiting volume (DLV). In general, the support on the chassis of the falling-object protective structures is rigid, so almost all impact energy is dissipated in the deformation of the cabin's strength structure. The present article proposes that most of the impact energy be dissipated in the post-critical deformation of some dissipative removable supporting elements interposed between the chassis of the machine and the falling-object protective structure. In addition, the article proposes the numerical evaluation of the post-critical behavior of the analyzed mechanical system, using finite element analysis. The first part of the article briefly presents the main regulations in the field of falling-object protective structures, as well as the methods of approaching the evaluation of the performances imposed on these structures, with their advantages and disadvantages. Following are given the main aspects of the optimal modeling of structures for numerical analysis of their behavior, the particularities of numerical evaluation of post-critical behavior and some constructive solutions of dissipative elements. The final part of the article is dedicated to a case study and the author's conclusions.

1. Preliminary

The continuous expansion and diversification of construction and landscaping works have led to the registration of a significant number of accidents involving machines and equipment specific to such works.

Excluding accidents due to improper operation of the machine by the human operator, statistics show that there are accidents caused by other factors, of which the "predisposition" to accident of the work area has a special importance. In this context are mentioned the work areas that favor the machine rollover and / or the falling of objects on the machine cabin. The consequences of such
accidents are often serious, both in terms of the human operator and in terms of the subsequent functionality of the machine.

Due to the gravity of the aforementioned accidents, over time have been developed regulations and standards that require construction machines to comply with certain performance criteria regarding the protection of the human operator in case of such accidents. These regulations and standards are reference elements both in the design and manufacture of the strength structures of the cabins of construction machines, and in the evaluation of their protective capacity.

If we refer strictly to accidents due to falling objects on the machine cabin, according to [1], the protection of the human operator is ensured if following the deformation of the machine cabin, no component of it penetrates a conventional volume (deflection-limiting volume - DLV) that is an orthogonal approximation of a large male, seated operator wearing normal clothing and a hard hat, according to [2]. Thus, the cab and its strength structure constitute a mechanical protection system, called the *Falling-object protective structure* (FOPS).

According to [1], for FOPS are imposed two acceptance levels:

- Level I impact protection - impact strength for protection from small falling objects. Energy of the impact is regulated to 1365 J.
- Level II impact protection - impact strength for protection from heavy falling objects. Energy of the impact is regulated to 11600 J.

### 2. Methods for evaluating the protection performance of FOPS

The main method of evaluating the protection capacity of the human operator by a FOPS structure is the experimental method. Laboratory testing of such a structure is regulated by the standard [1].

Briefly, FOPS (with DLV inside) is fixed on the testing stand similar to the fixing mode on the earth-moving machine chassis and is hit by a standardized test body having a geometric shape and mass depending on the level of protection imposed. The test body is freely falling from the height required by the insurance at the time of impact of the regulated energy, corresponding to the level of protection at impact imposed.

Although, like any experimental test, experimental testing of FOPS provides experimental results with a good level of accuracy, however the experimental testing of FOPS has some disadvantages, the most important of which are the following:

- The cabin and the machine on which it is mounted form a mechanical system that has a rigidity that is difficult to reproduce on the experimental testing stand. As a result, the actual impact energy dissipation capacity of the above mentioned mechanical system is different from the impact energy dissipation capacity that FOPS has, fixed on the testing stand. Consequently, the experimental results refer to a mechanical system approximately equivalent to the real mechanical system.
- In the vast majority of cases, even if FOPS "passes" the experimental test, it has plastic deformations, difficult to eliminate in order to restore FOPS to its geometric shape and initial dimensions. Therefore the tested structure is unusable after experimental testing.

In the context of the rapid growth of hardware and software resources of computing systems, the virtual testing of the protective structures of the earth-moving machinery can currently be considered.

Virtual testing of protective structures is done using FEA environments capable of simulating the behavior of mechanical systems in general, and therefore the behavior of structures subjected to mechanical shocks, as is the case of FOPS.

Basically, the virtual testing of a FOPS type structure uses a virtual model of it (usually generated in a CAD environment), not being necessary to manufacture the protective structure. Therefore, the main advantage of virtual testing of FOPS structures is given by the elimination of manufacturing costs of the experimental model of the evaluated structure. In addition, this approach gives the
engineer a great deal of freedom to modify all the defining parameters of the protective structure and to "experience" in a reasonable time the effects of the changes, in order to find the optimal constructive solution for FOPS.

However, the accuracy of the results obtained by virtual testing depends on several factors, the most important of which are the following:

- Adequate modeling of the behavior of materials attributed to the components of the protective structure, knowing that the material model is the main source of errors in the finite element method. In this context, it is noted that the material models associated with metallic materials lead to results with acceptable accuracy. Material models associated with composite materials, plastic materials, rubber-like materials etc. they must be carefully selected and possibly experimentally calibrated before use in finite element analysis.
- The optimal modeling of the studied problem (the shock caused by hitting FOPS with test body): constraints, loads, the type of finite element used in mesh generation etc.

3. Optimal modeling of structures for numerical analysis of their post-critical behavior

In general, if a given problem is considered, obtaining numerically, using the finite element method, of the results with the required level of accuracy is conditioned by achieving the optimal finite element model associated with the studied problem, ie that finite element model that leads to the relevant results with the imposed level of accuracy.

The optimal modeling with finite elements of the structures in order to analyze their behavior under the applied loads, involves the realization of several stages, of which the most important are described below.

3.1. Establishing relevant results.

Considering a structure with known boundary conditions (constarints and loads), the relevant results for the given structure are those results on the basis of which the most relevant observations and conclusions can be made on its behavior. The stage is of particular importance because it influences, among other things, the type of geometry associated with the structure and the type of finite element used to generate the mesh. In the particular case of evaluating the post-critical behavior of structures, the relevant results are the values of force and nodal displacements at different stages of the post-critical evolution of the structure because based on force-displacement plot, it’s possible to calculate the energy which is consumed in post-critical behavior.

3.2. Creating of the structure geometry

The geometry of the structure is the support of the mesh. The geometry of the structure, depending on its complexity, is created, either in the FEA environment (in the case of simple geometries), or in a CAD environment (in the case of complex geometries). Regardless of how the geometry is created, it must be a minimal geometry [3], ie. that geometry sufficiently simplified and not contributing to the unacceptable distortion of the relevant results to be obtained in finite element analysis. In the case of post-critical analysis, the geometry of the analyzed components, due to their geometric simplicity, is created in the FEA environment most of the times.

3.3. Material model

As mentioned above, the material model is the main source of errors in the finite element method. It follows the particular importance of adopting the appropriate model of material, ie. that model of material, simple enough and at the same time is able to describe the behavior of the real material [4]. In the particular case of the analysis of the post-critical behavior of components used for dissipative purposes made of structural steel (European standard EN 10025), the appropriate material model is a bi-linear model with isotropic hardening. Moreover, these steels, up to the yield stress, have negligible
deformations in relation to the deformations recorded after exceeding yield stress. As a result, a deformation-free bi-linear material model can be adopted until the yield stress is reached.

3.4. **Analysis and modeling of the interactions**
Most of the time the components whose behavior needs to be studied are part of component assemblies. Current FEA environments allow the analysis of component assemblies, but this approach is not always appropriate, especially when it is desired to analyze the behavior of only one component as a whole (target component). In such cases, it is much more productive to analyze only the behavior of the target component, because the associated finite element model is much simplified, thus consuming few resources and leading to results with similar accuracy, if not even better, with the accuracy of the results obtained by modeling the whole assembly of which the target component is a part. Obtaining veridical results in this way requires the correct modeling of the interactions of the target component with the related components as a whole. In this context, the components whose functional role is to dissipate impact energy through post-critical behavior are rigidly connected, both with the component that transmits the shock and with the one on which it rests. At the same time, the rigidity of the dissipative component is much lower than the rigidity of the components with which it is in contact. As a result, the components with which the dissipative component is in contact can be simulated in the finite element model by rigid surfaces, rigidly connected to the dissipative component.

4. **Constructive solutions**
The main aspects considered when adopting constructive solutions for dissipative components through post-critical behavior are briefly presented below.

The material of the dissipative elements must ensure a post-critical deformation of them, so that it is possible to dissipate the impact energies specific to the accepted levels imposed on the FOPS type protective structures. In this context, if the dissipative elements are made of steel, it must be a steel with good ductility (including at negative temperatures). Obviously, other materials can also be considered, but before they can be used, it must be checked whether their properties meet at least the ductility conditions imposed in [1].

The geometric shape of the dissipative element must ensure predictable post-critical deformations, so that, in turn, the FOPS displacement is predictable. One of the geometric shapes that ensures a good predictability of the post-critical deformation is the prismatic, tubular geometric shape. The result is dissipative elements made of pipe, preferably square pipe and circular pipe (common semi-finished products). In addition, the predictability of post-critical deformation as well as the value of its starting force can be influenced by making holes in the side walls of the dissipative element.

Because after the post-critical deformation of the dissipative elements, their reuse is compromised, the constructive solutions adopted must ensure their removability. As a result, it is necessary to consider a fastening system that allows the disassembly of the dissipative element (figure 1).
Figure 1. Removable attachment of dissipative elements between FOPS and chassis.

In order for the dissipative elements to ensure the required protection, it is necessary to mount them at the FOPS base, below the nodes where the vertical members with strength role of the structure converge (figure 1).

In addition to the above, the constructive solutions adopted for dissipative elements through post-critical behavior must ensure their achievement with low technological effort and low costs.

5. Case study

The case study is illustrative and has the following objectives:

- Approaching with the finite element method the study of the post-critical behavior of a dissipative element.
- Determining the energy dissipated by the dissipative element during the post-critical behavior.

The dissipative element is considered to be made of 80x80x2 square pipe. The length of the dissipative element is 300 mm.

The material of the dissipative element is the structural steel S235, EN 10025-2. The mechanical properties of S235 steel, necessary for finite element modeling of the problem, are given in table 1.

| Mechanical property                  | Unit  | Value  |
|--------------------------------------|-------|--------|
| Density, $\rho$                      | kg/m$^3$ | 7850   |
| Modulus of elasticity, $E$           | MPa   | 210000 |
| Poisson's ratio in elastic range, $\nu$ | -   | 0.3    |
| Yield stress, $\sigma_y$             | MPa   | 235    |
| Ultimate stress, $\sigma_u$          | MPa   | 360    |
| Ultimate strain, $\varepsilon_u$     | -     | 0.2    |

If a bi-linear material model is accepted, the strain corresponding to the yield point is:

$$
\varepsilon_y = \frac{\sigma_y}{E} = \frac{235}{210000} \approx 0.001 \ll 0.2 = \varepsilon_u
$$

The result in (1) justifies the statement made in paragraph 3.3, regarding the model of material associated with structural steels, EN 10025-2. Figure 2 shows the adopted model for S235 steel.

![Figure 2](image)

Figure 2. The adopted model for S235 steel.

The dissipative element is considered to be rigidly fixed to the ends, both on the car chassis and on the FOPS (figure 1). In the event of a shock, FOPS has vertical displacement, axially compressing the dissipative element. If the shock load leads to a force exceeding the elastic buckling limit, the dissipative element has a post-critical behavior.
In the case study, the finite element analysis of the post-critical behavior corresponding to the 165 mm vertical displacement of the upper end of the dissipative element is proposed (figure 3).

In order to simplify the finite element model associated with the formulated problem [5], in the FEA environment used were modeled two rigid flat surfaces connected with the dissipative element, at its ends, having the role of simulating FOPS, respectively the car chassis.

The fully constraint of the lower plane, respectively the vertical displacement of the upper plane were imposed by means of reference points defined in these planes (figure 4).

It should be noted that the imposed displacement is based on the condition that its sum with the vertical deformation of the protective structure (FOPS) does not exceed the distance between the vertical plane of the FOPS and the vertical plane of the DLV. Anyway, in a real situation not only one dissipative element is used, but several and not necessarily with the dimensions taken into account in the case study. In this context, the case study is illustrative only in terms of the high ability to dissipate energy through post-critical behavior of such dissipative elements.

Figures 5 and 6 show the von Mises stresses distributions in the dissipative element at the half of the imposed displacement, respectively at the end of it, obtained after solving the numerical calculation model associated with the finite element model.
In the context of increasing the passive protection of the protective structure (FOPS), one of the most relevant results of the analysis of the post-critical behavior of the dissipative element is the value of energy dissipated by it, during its post-critical evolution.

Considering the upper reference point (figure 4), for a certain displacement of it during the post-critical behavior \( z \), a vertical force \( F_z \) is required, which causes the deformation of the system at that moment. Figure 7 gives the \( F_z \) curve, corresponding to the post-critical evolution of the dissipative element considered in the case study.

![Figure 7. Variation of the \( F_z \) force during the postcritical behavior.](image)

Noting with \( \Omega \) the area between \( F_z \) curve and abscissa, the energy dissipated by the dissipative element, during its post-critical behavior is:

\[
U = \Omega \times 10^{-3} \approx 14970\text{J}
\]  

6. Conclusions
The post-critical behavior of some strength parts offers important capabilities to dissipate the energy of mechanical shocks. As a result, the removable dissipative elements, interposed between FOPS and the car chassis, correctly designed and manufactured, lead to a considerable increase in the passive protection provided to the human operator.

The use of removable dissipative elements, interposed between FOPS and the chassis, is preferable to the use of cab protection guard frames, because the dissipative elements are cheaper and require a simple fabrication technology.

The numerical evaluation of the post-critical behavior of the dissipative elements (practically, a virtual experiment) offers the advantage of studying several solutions in a reasonable time, which creates the premises for optimizing the geometric shape and their dimensions.

References
[1] EN ISO 3449 Falling-object protective structures. Laboratory tests and performance requirements Earth-moving machinery
[2] EN ISO 3164 Laboratory evaluations of protective structures. Specifications for deflection-limiting volume Earth-moving machinery
[3] Dumitrache P and Goanță A M 2017 Validation by numerical simulation of the behaviour of protective structures of machinery cabins subjected to standardized shocks IOP Conf. Ser.: Mater. Sci. Eng. 227 012041
[4] Yang Z 2019 *Material Modeling in Finite Element Analysis* (Florida: CRC Press)

[5] Ataei H, Mamaghani Mo 2017 *Finite Element Analysis Applications and Solved Problems Using Abaqus* (California: CreateSpace Independent Publishing Platform)