RESEARCH ARTICLE OPEN ACCESS

Development of a Model for the Simulation of ROPS Tests on Agricultural Tractors Cabin: Numerical Models and Experimental Verification

[1,2]Sergio Baragetti, [3]Franco Robolotti, [4]Fabio Rota
[1]Dipartimento di Ingegneria Gestionale, dell'Informazione e della Produzione, Università degli Studi di Bergamo, Viale Marconi 5 – 24044 Dalmine (BG), Italia.
Department of Management, Production and Information Engineering, University of Bergamo,Viale Marconi 5 – 24044 Dalmine (BG), Italy.
[2]GITT – Centreon Innovation Management and Technology Transfer, University of Bergamo, Via Salvecchio 19– 29129, Bergamo, Italy.
[3]R&D Calculationdepartment, SameDeutz – Fahr Italia S.p.A., from 2002 to 2008,Viale F. Cassani 15 – 24047 Treviglio (BG), Italia.
[4]Dipartimento di Progettazione e Tecnologie, Università degli Studi di Bergamo, from 2001 to 2006,Viale Marconi 5 – 24044 Dalmine (BG), Italia.

Abstract
It is here proposed a methodology for simulation of ROPS tests (ROPS = *Roll Over Protective Structure*) of agricultural tractor cabins. The work is based on the resolution of this problem through the use of the finite element method. In order to limit the number of nodes of the model and thus to speed up the resolution,a two-dimensional finite elements model has been chosen. The method presented here solves with relative ease, even very complex structures. There are also simplest methods in literature where specially made software is based on the finite element method for simulating approval tests on ROPS structures. In this case,codes developed just for this purposeare available, and therefore very simple to use and characterized by a high speed of preparation of the model following the definition of a small number of parameters. On the other side these are codes designed for structures having a specific geometric shape and in which the user is not free to set all the parameters existing in commercial software for the structural calculation, and are not very suitable in case of complex or not conventional structures. The methodology proposed by the authors instead, although not automated, allows simulating any type of structure in acceptable times. The results were validated by full scale experimental tests. Through the interpretation of the results it is possible to identify which areais the most critical for the structure and evaluate any change, something which is not easy to do through expensive tests.
Keywords: agricultural tractors; ROPS; FEM models; OECD codes.
References in this article may be made as follows: S. Baragetti, F. Rota and F. Robolotti., (2015), "Development of a model for the simulation of ROPS tests on agricultural tractors cabin" <name magazine>, Vol . <n ° volume>, No. <No.>, pp. <Pages>.

I. Introduction

The focus on the protection structures for agricultural tractors in case of rollover (commonly referred to as ROPS, *Roll Over Protective Structure*) is relatively recent, and it depends on releasing, by different countries, of regulations regarding the requirements to equip tractors of such security elements.

On the same time even some international organizations have promulgated guidelines, codes and standards about protective structures for tractors, such as ISO codes, issued by the homonymous organization, and OECD codes, promulgated on behalf of the EEC [1-5].

From the structural point of view the cabin constitutes a safety device for an agricultural tractor also able to support the entire tractor in case of overturning. Therefore, this element is crucial to the safety of the driver and must meet specific requirements that must be checked before the start of production of the cabin or of a substantial modification through appropriate tests.

By analyzing data of the approval tests carried out during the years 1994 - 1998 at the Experimental Centre of the Universityof Agriculture of Bologna, it was found that in 28% of cases the test ended with a failure.

Most of the unsuitable structures were then modified and resubmitted, and in several cases failed one or more tests later.

It should be emphasized that each verification, in general, requires a considerable effort from the company for manufacturing the prototype, and that especially the failure of a test often involves a significant delay in the series production.

However the effort invested by the companies in creating serious experience with appropriate calculation methods for mechanical design was not always adequate.

Such attempts can be classified in accordance with the principle on which they are based. We can distinguish:

- software specially made for the simulation of approval tests on ROPS structures: these are very simple to use and are characterized by a high speed of preparation of the model following the definition of a small number of parameters. On the other sidethese are codes designed for structures having a specific geometric shape and in which the user is not free to set all the parameters existing in a commercial software for the structural calculation. Therefore the software is not very suitable in case of complex structures or unconventional design;
- commercial programs for the structural analysis of nonlinear models.About these packages it is worth recalling that these are general purpose applicationsso,notwithstanding they can treat a much larger number of cases leaving more freedom to user, they requireadequate technical and scientific know-how, and often a significant commitment of time and money.

Comparing a considerable industrial interest in the development of methods for the design of ROPS a limitedcontributions from the world of research has to be evidenced. From the national and international scientific literature [6-13] it emergesthat most of the publications were published before 2002.Furthermore they are related almost exclusively to structures modelled with a “frame” scheme, so easily simulated by one-dimensional finite elements.

II. Scope of the Work

Purpose of the present study is to define a methodology to simulate, through the finite element method, the approval static test of ROPS cabin for large tractors (~ 300 CV) as per OECD codes.
From a result point of view there is also the target of reaching values, as regards the requirements of the OECD codes, that differ from the experimental data at the most 10%.

III. The cabin

The cabin object of this paper is the model SAME Deutz-Fahr GC8, approved for tractors with mass up to 9000 kg (Figures 1, 2, 3).

Figure 1: The overall cabin

The structural part is constituted firstly by a series of plates that are the base of the cab itself.

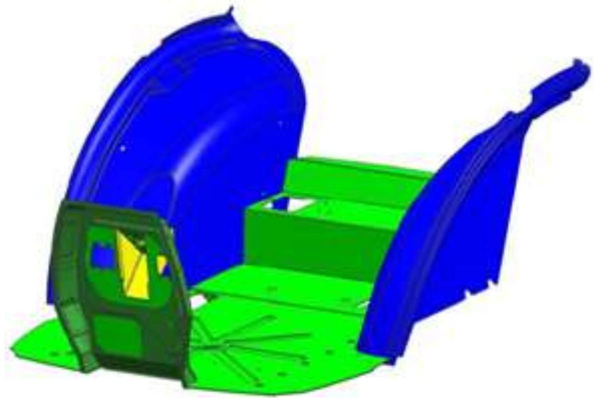

Figure 2: Base of the cabin

The rest of the cabin is constituted by a series of metal profiles having the function to realize the protective metal cage within which the driver can find a shelter in case of overturning of the tractor.

Figure 3: Structural part of the cabin

These profiles, reinforced through the use of other profiles and support plates, are joined together by a series of nodes made of steel. The continuity between the various elements is guaranteed by a series of continuous welds and tack welds.

Fixing of the cabin to the chassis of the tractor is made in four points, two front and two rear, through appropriate components with an elastic element, denominated *silent-block*, having also the function of damping the vibrations that the tractor transmits to the cabin itself (Figure 4).

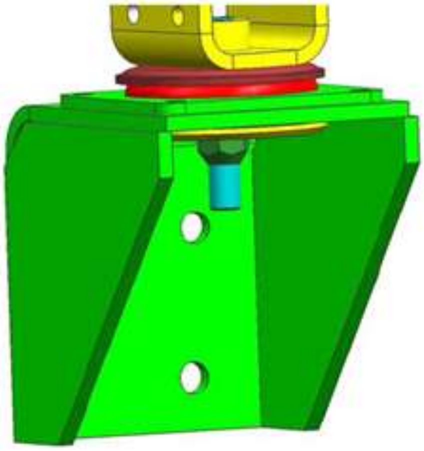

Figure 4: Cabin fixing point with silent-block

IV. The approval test

The reference, with regard to the approval of this type of protection structures, is theOECD code 4.

According to OECD code 4the cabin is subjected to the action of hydraulic jacks, which, according to a precise sequence, must act in well-defined points of the cabin, deforming it permanently. Given the low speed at which the actuators operate (v ≤ 5 mm/s), the test is considered quasi-static.

As already mentioned this test consists in four load cases to be carried out all on the same cabin and following a defined order:

- *Longitudinal load:* in this case the actuator acts in the horizontal plane; this load phase is stopped when:
• the energy absorbed by the protective structure is greater than or equal to the energy required (1.4 M [J]where M [kg] is the mass of the un-ballasted tractor);
• the protection structure occupies the clearance zone or leaves it unprotected (Figure 5).

Figure 5:Longitudinal load

- *Rear vertical load*: here the structure is subjected to a load belonging to the vertical plane; to pass the test the structure must sustaina crushing force F = 20 M [N] for five seconds without any detectable movement of the protective structure (Figure 6).

Figure 6: Rear vertical load

- *Lateral load:* as for the longitudinal load, also in this case the actuator acts in the horizontal plane; this load phase is stopped when:
• the energy absorbed by the protective structure is greater than or equal to 1.75 M [J].
• the protection structure occupies the clearance zone or leaves it not protected (Figure 7).

Figure 7: Lateral load

- *Front vertical load*: here the structure is subjected to a load belonging to the vertical plane; to pass the test the structure must sustaina crushing force F = 20 M [N] for five seconds without any detectable movement of the protective structure (Figure 8).

Figure 8: Front vertical load

The integrity of the clearance zonesis requiredin all load phases, a safety volume inside the structure that depends on the position of the seat and the steering wheel and which must never be invaded by the security structure, nor be left without protection (Figure 9a,b,c).

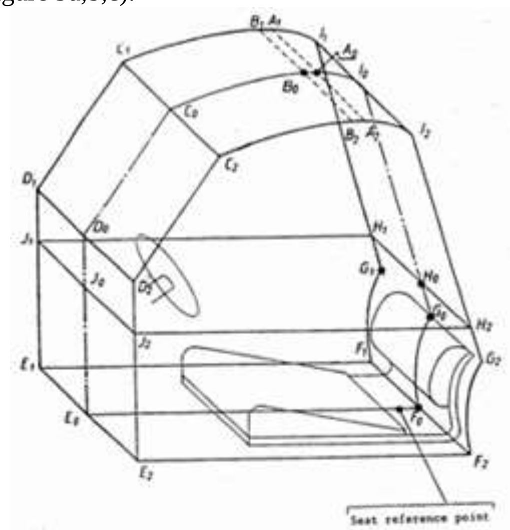

Figure 9a: Clearance zone

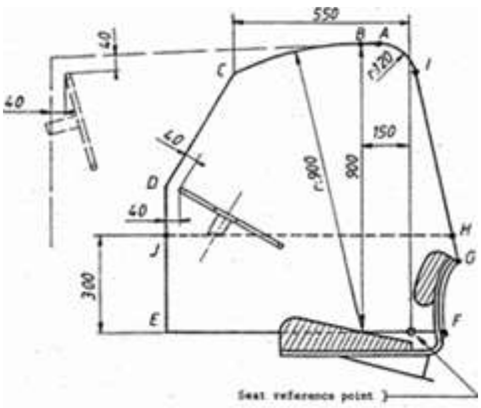

Figure 9b - Clearance zone (dimensions in mm)

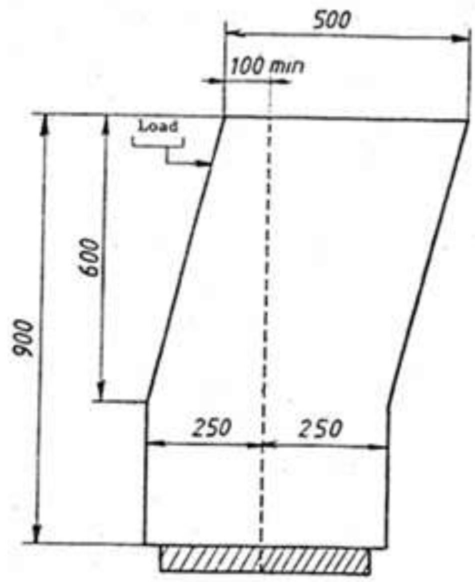

Figure 9c - Clearance zone (dimensions in mm)

V. Framework modelling

To simulate the approval test a commercial software for the nonlinear structural analysis was chosen. In particular the nonlinear solver MSC.Marc® was used. As pre/post-processor the software MSC.Patran® 2005 was used, as this pre/post-processor is compatible with different solvers, allowing to transfer the model from a solver to another one without having to start over again, once prepared. This makes it possible to assess the "sensitivity" of the model too.

The first target is to prepare a simplified geometric model from which the finite element model of the cabin will be developed. The starting point is the three-dimensional geometric model of the cabin.

All non-structural componentsare deleted from tha model;these are the components of the protection structure that are not subjected to the approval test, such as panels, the dashboard, windows, the exhausts system, the electrical system, plumbing, and the non-structural part of the roof.

Despite being part of the structure of the approval test, also the steering wheel and the seatwerenot considered. The main task of these elements within the approval test is indeed to assist in defining the clearance zone.As a first approximation it can be considered negligible their contribution to the strength of the cabin.

All the components of non-permanent connections between the various parts were also eliminated; such elements, always in order not to unnecessarily increase the complexity of the model, weremodelled later.

Finally the geometrical parts constituting the *silent-block*were also eliminated since, as will be seen better later on, the behavior of such components was simulated by means of special elements, always in order to obtain a model as simple as possible but that allows giving results having good reliability.

Due to the considerable presence of laminated suitably shaped components having significant structural contribution, as well as a series of hollow profiles, the decision was to use two-dimensional finite elements. This modelling allows describing with a good approximation also the local effects, especially with regard to nonlinearity; the change of the shape and size of the cross section of the components was considered too.Such aspect is completely ignored with a one-dimensional elementsapproach. The use of two-dimensional elements may be essential where important phenomena of shape instability of the components may happen, and where also several parts of metal sheet as well as profiled have to be simulated. Furthermore, the use of two-dimensional finite elements, if compared tothethree-dimensional ones, allowssaving a huge amount of computer memory and resources and speed up the numerical procedure.

Figure 10: FEM model of the cabin

At this level the model is not yet completed, as there is still to simulate the presence of some components belonging to the structure. The FEM model of the cabin is shown in Figure 10.

VI. Material definition

Firstly, in order to define which kind of materials include in the model, it is important to understand which are the materials used in the structure.

The model will greatly deform even in the plastic range. Therefore it is not possible to synthesize the behavior of all the materials with one single steel, nor to assume the hypothesis of the perfectly elastic material. The plastic behavior of the materialhas to be modelled, and also several different types of steel are necessary, given the considerable differences in behavior in the plastic field.

On the other hand it is important to simplify also this stage of modelling, using a minimum number of materials, especially considering that the differences between the materials are minimal. It is therefore essential to establish how many and which materials have to be modelled.

The first step in MSC.Patran®is to define these materials in terms of true stress – true strain, as this is the curve requested from the software.

The relations between engineering dimensions and truedimensions are the following ones:

$$\sigma_{true} = \sigma_{eng} (\varepsilon_{eng} + 1)$$
$$\varepsilon_{true} = \ln (\varepsilon_{eng} + 1)$$

An approximation of the post necking portion of the curve can be achieved according to the construction of Considere.Such construction is created starting from a diagram that plots the true stressversus the engineering strain.

Tensile tests of the different materials used in the structure were performed.

The specimen (Figure 11a)werecut directly from the metal sheetin accordance with UNI EN 10002-1 and UNI EN ISO 377.

Figure 11a: Specimen for the tensile tests

Unfortunately for this geometrya corrective method for the determination in engineering terms of the post necking curve does not exist. It was therefore decided to simulate, using the finite elements method, the tensile test of the specimen in order to obtain themissingexperimental-numericalcorrelation.

Due to the symmetry in geometry and load of the problem only one quarter of the specimen was modelled (Figure 11b).

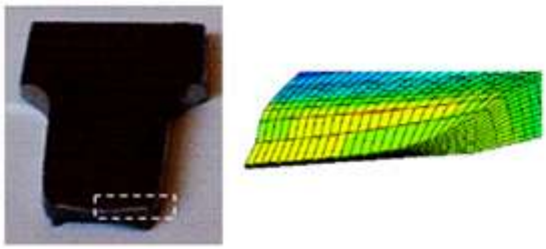

Figure 11b: FEM model of the specimen

This way it was possible to have the information required to build the numerical model representative of the materials.

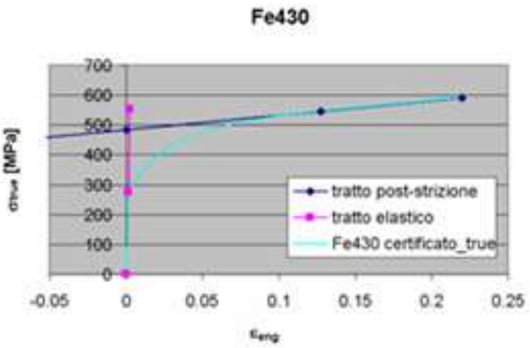

Figure 12: Material stress-strain curve

Given the quasi-static nature of the tests, in the present study the effects of embrittlement of metals, related to the speed of deformation of the material were not taken into consideration.

It is therefore exclusively considered an isotropic hardening, fully described by the stress-strain curve of the material. The effects of kinematic hardening were omitted as in this case each phase of the load is constituted by a single cycle and not by cyclic loads over time. The von Mises criterion of resistancewas assumed, due to the ductility of the materials,in order to determine the yield strength. A stress-strain curve is shown in Figure 12.

VII. Defining the elementsproperties

In order to fully characterize the behavior of the two-dimensional elements it is necessary to define their characteristics. Consistent with the possibilities offered by MSC.Marc®thick shell type elements wereused. For the items associatedwith this type of elements the normal stress out of the plane of the elements wasignored. They are particularly suitable in the case of bodies with small thickness.

VIII. Modelling of welded joints

In the protection structure there are three different types of welded joints:
- *Tack welds*, mainly used to connect sheet metal components;
- *Spot welds*, to connect the reinforcement sections to the respective mast and crossmember;
- *Continuous welds*, to join together all the other components.

To simplify the preparation of the model and avoid convergence problems caused by possible overstressing and high local plasticization, the tack welds were modelled like the continuous welds. It is so accepted a possible increase in the structure stiffness due to the higher degree of constraint provided by continuous welds than tack welds.

The continuous welds and tack welds are modelled with a series of two-dimensional linear elements that allow connecting between them the correspondingcomponents. The material property of the relative components are then assigned to these weld elements.In order to simplify the model thefollowing items were not modelled:
- *The actual geometry* of the welds;
- *The presence of the heat affected* zone on the weld and in the base material near the weld;
- *Possible damage phenomena under stress of the welding area*. It may happen that the welds are not performed correctly or that are placed between components having a very different thickness that means different possibility of absorbing heat during the welding phase. In the latter case it may happen that the base material does not melt, creating a gluing instead of a weld. This could result, during the test phase, in gaps between the welded parts, decreasing the tensile-deformation response of the structure.

The welding points in the structure were instead modelled with one-dimensional linear elements that locally connect a node of the reinforcement to a node of the reinforced component in the position where the welding spot is located.

Even for these items the material properties are requested. These elements are considered as standard straightbeam elements;they are very shortif compared to the characteristic size of their section, and for simplicity of calculation a perfectly elastic behavior of the material was selected.

IX. Modelling of plugs

Together with spot welds also plugs are used in the protective structure. If the main task of the weld is to enhance the stiffness of themast and cross members, the use of the plugs is necessary to keep in position the reinforcing elements, but also to replace the welding point where itcannot be realized.

Like thespot welding even the plugsweremodelledwith one-dimensional linear elements that locally connect a node of the reinforcement to a node of the reinforced component in the position where the plug is located.Even in this case they are considered as standard straight beam elements, with a perfectly elastic material behavior.

X. Modelling of bolted joints

The bolted joints were modelled with two different criteria depending on the different modes of deformation of the areas in which these junctions operate.

Where the bodies joined by such joints are rigid enough to assumethey deform as if they were a single one, the bolted jointisn’t modelledwith an FE element but witha Multiple Point Constraint (MPC). The MPCs are constituted by relations that allow relating displacements and rotations of different nodes. For this model the MPCs "rigid" type were used: in this way it was possible to impose to one or more “dependent” nodes to assume at any instant the same displacements and rotations of another "independent"node.

Where the two bodies joined by bolted joints can deform quite differently, a greater degree of freedom of nodes movement is modelled. Therefore in this case bolted joints were modelled by one-dimensional linear elementsrather than through MPCs. These elements areconsidered as standard straight beams with a perfectly elastic material behavior to simplify the model, given their higher stiffness in comparison to the connected metal sheet.

XI. Modelling of the silent-blocks

The silent-blocks are used to connect the cabin to the rest of the tractor frame and for dampening the vibrations transmitted from the frame to the cab itself. All the protective structures are equipped with four identical silent-blocks, one for each connection. The silent-block is physically constituted by (Figure 13):

- A rubber component having a damping function;
- A link plate of the rubber component at the lower plate;
- A connecting pin of the rubber element to the upper plate;
- A top end stroke plate, placed on the top of the rubber element, acting as end stroke in the case of loads acting on the plate headed down;
- A bottom end stroke plate, placed under the rubber element, acting as end stroke in the case of loads acting on the attack directed upwards.

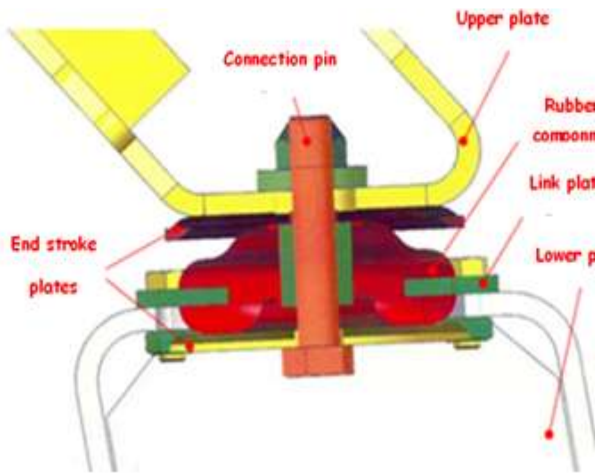

Figure 13: Silent-block

In this simulation, the target is to model as closely as possible this component, because it has to bear all the stresses that are transferred fromthe cab to the tractor. The behavior of this component can greatly affect the stiffness of the entire structure. However the model of the silent-block must be as simple as possible, in order not to unnecessarily burden the analysis.

For this purpose the rubber component is replaced with elements having its own elastic characteristic in the axial and radial directions. Three mono-dimensional elements were created for each silent-blockhaving the end points at the intersection of the axis of the silent-block with the upper plate and with the lower plate.

To define the properties to be assigned to these elements, the elastic characteristics of axial and radial components of the rubberwere used. The radial and axial silent-block stiffness characteristics are shown in Figure 14.

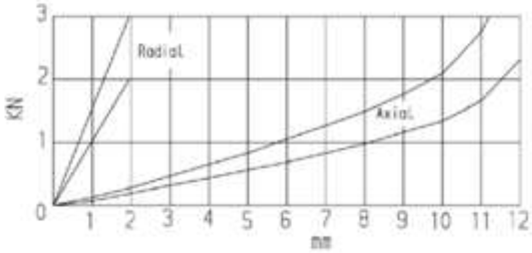

Figure 14: Silent-block stiffness characteristics

Using these characteristicsthe three elements are modelled as linear springs, for simplicity of the model and given the good approximation that however is possible to achieve.

To connect the ends of these elements to the upper and lower plates,a series of two-dimensional linear type TRIA elements were used(Figure 15).

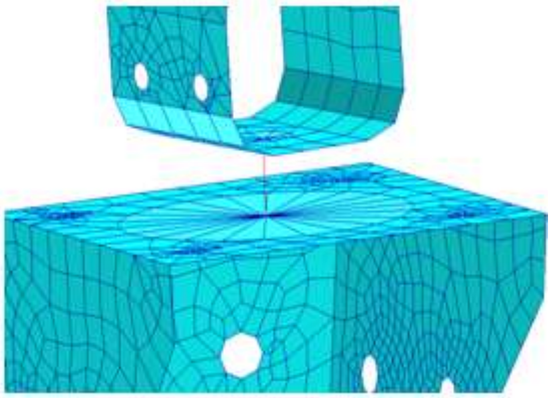

Figure 15: Silent block axial FEM model

For the modelling of the end stroke two different approacheswere used: one forthe maximum displacement allowed between upper and lower plate in the radial direction and a second one for the displacement in the axial direction and the rotations admitted.

Let's start with the allowed movements in the radial direction. This constraint wascreatedby modelling for every silent block a series of one-dimensional linear elements having the same extremes of the elements that simulate the rubber component. These elements were defined in the category of gap fixed direction. The items in that category do not have structural properties, but allow limiting the relative displacement of their end nodes in a given direction.

As regards the movements allowed in the axial direction the maximum permissible relative displacement in the axial direction and rotations between upper and lower plateare defined through experimental tests and the theory of circular plates and shells.

Once the data are available it is possible to better define and model the end stroke. It was decided to represent these components by means of two rectangular surfaces for each silent-block made with two-dimensional linear QUAD type elements. These surfaces, realized in such a way as to have roughly the size of the end stroke plate that replace, were appropriately positioned one above and one below the bottom of each plate. Finally each edge was boundedto a node belonging to the lower plate with an MPC, in order to rigidly connect end stroke and lower plate. Given that the end stroke is a component of the lower plate, each of these MPC nodes belonging to the end strokeisa dependent node, while the node belonging to the lower plate acts as an independent node (Figure 16).

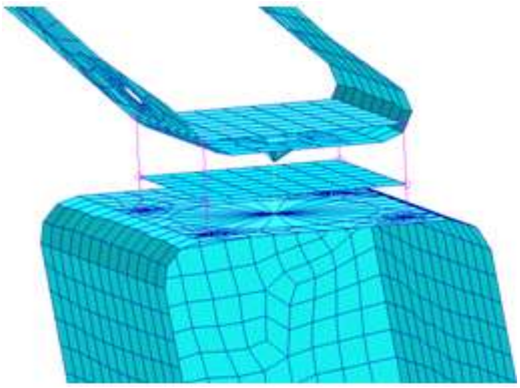

Figure 16: Silent block complete FEM model

The elements that constitute the end strokeweremodelled as two-dimensional elements anda perfectly elastic material property was assigned.

XII. Contactsmodelling

A fundamental importance has the correct definition of the contact elements. In finite element modelling, in order to simplify the model, it wasnot defined any type of contactbetween the elements.In this way elements belonging to different components, or even belonging to the same component, should not be in contact when one isnear the others, but penetrate as if one was not aware of the existence of the other. In order to overcome this problem the solver offers the possibility to define groups of geometric entities or finite elements, said contact bodies, for which the solver has to verify in each calculation step if these bodies are in contact or not. For these bodies, in this case, no friction behavior was defined, because it is was considered to be negligible for our purposes; finally, it would be a further complication of the model, as well as being another data to evaluate.

In the finite element model the elements were left free to penetrate between them, unless they were associated with special properties that made them contact bodies. However, since this type of approach may add considerably complication to the model, often alternative solutions were used to solve this problem.

The definition of a contact in a finite element model implies that, at every step of the calculation,the possible penetration is checked for every node of each contact body with every other node of every other contact body, thus obliging the solver to verify a large number of relationships between pairs of nodes.

If it is easy to identify the pairs of nodes that will be in contact with each other because of the stress agents, although they are not in contact at the beginning of the analysis, it is possible to locally define a relationships between pairs of nodes instead of contacts between interested contact bodies.This will help to decrease significantly the number of relations to be verified at every step of the calculation. Consequently required time and computing resources willdecrease significantly.

These relationswere made by placing gap elements between the pairs of nodes. This gap elementsrestrict the relative displacement while approaching of the two end nodes.

Furthermore, in order to simplify as much as possible the model, these relationships were defined locally only where their absence would lead to a deep and relevant penetration between the parts(relevant for the final response of the structure) thus leaving all the otherelements of the model free to penetrate.

XIII.Modelling of the clearance zone

The clearance zone is added to the model not to increase the stiffness characteristics of the cabin, but to verify that during the loading cycle the protection structure does not invade this volume.

For this purpose theclearance zonewas reproduced following the guidelines provided by the OECD Code 4, using two-dimensional linear FEs.

To perform the analysis virtual properties were assigned to these elements.

The so modelled clearance zone has to be connected to the cabin, so that it follows the deformation during the load cycle. In order to achieve this effect without affecting the stiffness of the cabin it was decided to connect via an MPC a node of the clearance zone to a single node belonging to the floor.

XIV. Creating constraints

Once the Finite element model was completed it had to be connected to the ground. Since:

- The lower plates of the silent blocks are connected with boltsto the frame of the tractor;
- The frame of the tractor is considerably more rigid than the cabin;
- The frame should not affect the approval test;the lower plates of the silent blocks are directly fixed to the ground.

Since even the lower plates, having high stiffness, deform a little, the contact between the lower plates and the frame was not modelled but the cabin was constrained so that any kind of shift and rotation of the nodes constituting the edges of the holes was avoided (Figure 17).

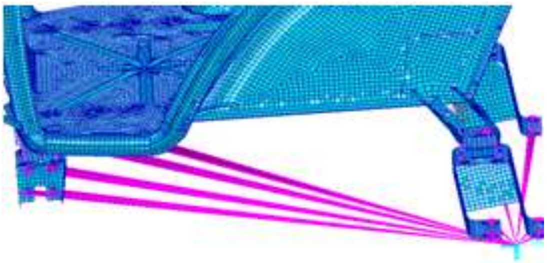

Figure 17: Cabin constraints

XV. Modelling of the pushers

The pushers are the external elements that apply the forces to the cabin during the approval test. In order to achieve more reliable results they were directly modelled instead of using forces applied to the structure.

The pushers were modelled with rigid elements.This way the stiffness of the pushers did not affect the final deformation of the cabin. To have a better correlation with the experimental tests the final results of the experimental testswere analyzed in order to remove the contribution of the stiffness of the external frame of the pushers.

This way the OECD Code 4 requirements were fulfilled, since within the standards the pushers are defined as rigid, but without defining any limit value.

These rigid contact bodies were controlled with an enforced displacement during the analysis.

The above way to model the pusherswas easy and did not add too much complication to the model.The limit was that the movement of the pusher is totally controlled andit is not possible to leave freetherotationaldegrees of freedom that areneeded in order to simulate the presence of the universal joint in the real test structure that allows pushers adapting their orientation to the structure.

Due to this limit the pushers must be correctly positioned from the beginning and are not free to rotate, because this could lead to mistakes in the simulation of the approval test (Figure 18).

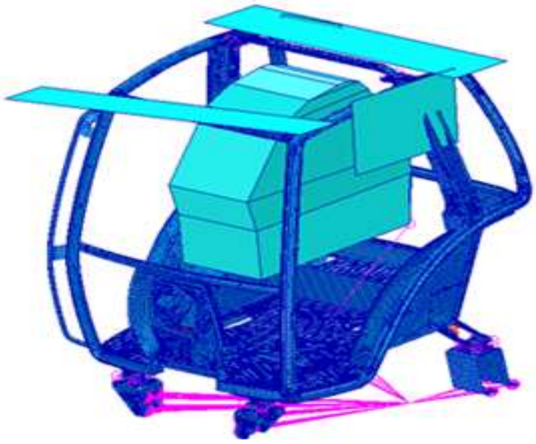

Figure 18: Pusher and clearance zone

XVI. Analysis of the results

The cabin is composed of 76,378 nodesand a total of 79,443 elements. The time required for the calculation of the simulation related to the whole load cycle is about 55 hours, where the computer used is equipped with a dual processor Intel Xeon 3.6 GHz and 4 GB of RAM memory.

Once the analysiswas completed the results were compared with the experimental data.

This was done from two different points of view:

- Comparing the diagrams force-displacement ofthe experimental test with data obtained from the numerical simulation;
- Comparing the respective final deformed structure.

The comparison between the experimental diagrams force-displacement and the simulated ones is very important because:

- It allows on one hand to numerically evaluate, from the point of view of the results, the goodness of the analysis performed;
- On the other hand it is possible to sense the differences between experimental evidence and simulated tests and then determine any defects presented by the model compared to the experimental tests (Figure 19).

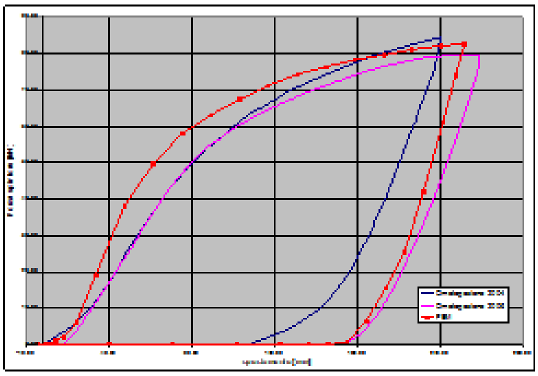
Figure 19: force–displacementdiagrams

In this case the deviation between the values required by the approval code, obtained from the numerical simulations and the real tests, is equal to:
- 5.3% incase of longitudinal load;
- 6.2% for the vertical load;
- 23.4% for the lateral load;
- 7.3% for the front vertical load.

As already mentioned the comparison between the actual and simulated deformed configuration isvery important. Although this comparison is morequalitative than quantitative, this analysis is very important because it allows assessmentof the comparison of the deformation of the modeland the real structure Furthermore this analysis allows deciding what to do to improve the model. Once the modelwas validated, the analysis of its deformationallowedcheckingwhich are the critical areas in the various loading phases, and then where the protective structure has to be reinforced (Figures 20-27).


Figure 20: Comparison between the test and the FEM model


Figure 21: Comparison between the test and the FEM model


Figure 22: Comparison between the test and the FEM model

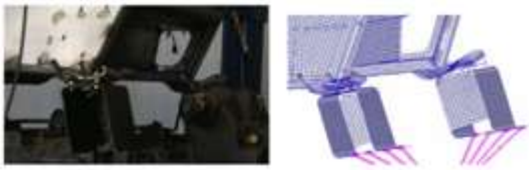
Figure 23: Comparison between the test and the FEM model


Figure 24: Comparison between the test and the FEM model


Figure 25: Comparison between the test and the FEM model

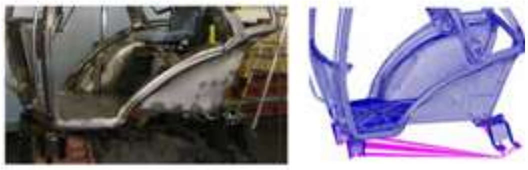
Figure 26: Comparison between the test and the FEM model


Figure 27: Comparison between the test and the FEM model

XVII. Conclusions

This work led to the definition of a methodology that proved capable of simulating, through the finite element method, the approval test of ROPS static structures type cabin for large tractors (~ 300 CV) under the OECD codes.

Compared to the experimental data the obtained values deviate, as regards the results required by OECD codes, of:
- 5.3% for pushing the rear;
- 6.2% for crushing the rear;
- 23.4% for the lateral thrust;
- 7.3% for crushing the front.

The methodology proved to beeffective despite the approximations adopted.In this sense it is necessary to point out some of the main changes that can refine the model without requiring additional resources and computational time, such as:

- A full set of experimental tests designed to fully characterize the behavior of the mounting silent block–plates;
- A full set of experimental tests in order to determine the true characteristics of the materials instead of determining them by extrapolating from those of other available materials;
- Modelling the tack welds as they are and not as if they were continuous welding to avoid unnecessarily stiffen the structure.

Having more available time and computing resources, it would be possible to make the following improvements to the model:

- Further improve the mesh in the areas with large deformations, in order to have a better response of the model in this respect;
- Model in more detail the silentblock;
- Introduce criteria of rupture, based for example on a maximum permissible deformation, for all the elements and in particular for the welding;
- Shaping pushers in such a way as to permit their free orientation when they come in contact with the cabin;
- Re-evaluate the way in which the reinforcements are joined to their respective profiles.

Various tests may also be performed to assess the sensitivity of the model to the solver used, varying parameters or adopting other solvers.

Bibliography

[1.] Code 4 – OECD standard code for the Official testing of protective structures on agricultural and forestry tractors (static test) OECD, 2005.
[2.] www.oecd.org, OECD official internet site.
[3.] La Sicurezza delle Macchine Agricole – Parte Generale, ENAMA, 2002.
[4.] La Trattrice Agricola a Ruote, ENAMA, 2002.
[5.] L'uso in sicurezza delle macchine per la lavorazione del terreno, ENAMA, 2002.
[6.] C.J. Chisholm, J.F. Parker, "The prediction of tractor safety cab deformation: Part I e II – A preliminarystudy",Nat. Inst. Agric. Engin., Silsoe, 1974.
[7.] A. Fabbri, "Sviluppo di un programma di calcolo, basato sulla tecnica degli elementi finitiper la progettazione delle strutture di protezione installate sulle trattriciagricole",Rivista di Ingegneria Agraria, 2000, n°3, pp. 176 – 188
[8.] A. Fabbri, A. Guarnieri, "Le ricerche sul comfort e sulla stabilità delle trattrici agricole",Convegno Nazionale AIIA, 2002.
[9.] A. Fabbri, S. Ward, "Validation of a Finite Element Program for the Design of Roll – Over ProtectiveFramed Structures (ROPS) for Agricultural Tractors",Biosystems Engineering, 2002, n°81, pp. 287 – 296.
[10.] J.R. Harris, V.H. Mucino, J.R. Etherton, K. Snyder, K.H. Means, "Finite element modelling of ROPS in static testing and rear overturn",Journal of Agricultural Safety and Health, 2000, n°6(3), pp. 215 – 225.
[11.] D. Pessina, J. Radovan, "Studio delle caratteristiche delle strutture di protezione dei trattori agricolitramite la modellizzazione a elementi finiti",Atti del XII Convegno Internazionale CIGR – AgEng, 1994, pp. 1442 – 1453.
[12.] J. Mangado, J.I. Arana, C. Jare´n, S. Arazuri, P. Arnal,"Design Calculations on Roll-over Protective Structures for Agricultural Tractors", Biosystems Engineering (2007) 96 (2), 181–191.
[13.] A.L. Guzzomia, V. Rondellia, A. Guarnieri, G. Molari, P.G. Molari., "Available energy during the rollover of narrow-track wheeled agricultural tractors", Biosystems Engineering (2009), 104, 318-323.