5th Youth Symposium on
Experimental Solid Mechanics

May, 10-13 2006
Púchov, Slovakia

STRENGTH ANALYSIS OF SAFETY CAGE

ANA PAVLOVIC[1*], MIROSLAV ZIVKOVIC[2], SNEZANA VULOVIC[3], GRUJOVIC NENAD[4]

[*] ***Faculty of Mechanical Engineering; Kragujevac University, Serbia and Montenegro, Sestre Janjic 6, 34000 Kragujevac,***
[1] ***e-mail: miakg81@yahoo.com***
[2] ***e-mail: zile@kg.ac.yu***
[3] ***e-mail: vsneza@kg.ac.yu***
[4] ***e-mail: gruja@kg.ac.yu***

ABSTRACT: Safety cage represent one of the most important part of equipment in sport cars. It has to be made in the way to absorb kinetic energy and on that way reduce contusion of driver. Initiation world standards of safety cage, in car production, in one of our companies were the main motivation for this work. During the race, cars with safety cage (common name: roll bar) have to be completely safety, especially in the case of turning over or crash. In this case we didn't have acceleration life test and any improve that structures is going to resist impact. Structures had to be modeled in CATIA software like a shell. Geometrical and material non linear analysis was done using software PAK [1]. Using displacement control method we got critical forces for structure. Those results gave necessary safety factor and strength, and acceptable results were obtained using material with better properties. Procedure that was used in this work reduced the number of experimental testing and their costs.

KEY WORDS: Modelling of safety cage, FEM simulation, strength analysis

1 METHOD FOR SOLVING NONLINEAR EQUATION

Methods for solving non linear equation can be classified in two basic groups: incremental (in steps) and iterative or *Newton's* methods. Usually it is used incremental-iterative procedure, with the stress integration and with calculation of the tangent constitutive matrix. According to the below table we see that the stress integration and calculation of the constitutive relations represent one of the key steps in an inelastic incremental analysis.

A. Initial conditions for step Δt

$$i = 0 \; ; \; {}^{t+\Delta t}U^{(0)} = {}^{t}U \; ; \; {}^{t+\Delta t}F^{(0)} = {}^{t}F \; ; \; {}^{t+\Delta t}K^{(0)} = {}^{t}K$$

B. Equilibrium iteration loop on the structural level

$$\mathbf{i} = \mathbf{i} + 1$$

C. Loop over integration points (stress integration)

Total strain $\;{}^{t+\Delta t}\mathbf{e}^{(i-1)} = {}^{t+\Delta t}\mathbf{B}^{(i-1)}\;{}^{t+\Delta t}\mathbf{U}^{(i-1)}$

known: ${}^{t}\sigma, \; {}^{t}\bar{e}^{P}, \; {}^{t}\mathbf{e}^{P}, \; {}^{t+\Delta t}\mathbf{e}^{(i-1)}$

unknown: ${}^{t+\Delta t}\sigma^{(i)}, \; {}^{t+\Delta t}\bar{e}^{P(i)}, \; {}^{t+\Delta t}\mathbf{e}^{P(i)}$

Constituitive matrix $\;{}^{t+\Delta t}\mathbf{C}^{(i-1)} = \dfrac{\partial\, {}^{t+\Delta t}\sigma^{(i-1)}}{\partial\, {}^{t+\Delta t}\mathbf{e}^{(i-1)}}$

Internal nodal forces $\;{}^{t+\Delta t}\mathbf{F}^{(i-1)} = \sum {}^{t+\Delta t}\mathbf{B}^{(i-1)}\;{}^{t+\Delta t}\sigma^{(i-1)}\Delta V$

Stiffness matrix $\;{}^{t+\Delta t}\mathbf{K}^{(i-1)} = \sum {}^{t+\Delta t}\mathbf{B}^{(i-1)T}\;{}^{t+\Delta t}\mathbf{C}^{(i-1)}\;{}^{t+\Delta t}\mathbf{B}^{(i-1)}\;\Delta V$

Increment of displacements and total displacement

$${}^{t+\Delta t}\mathbf{K}^{(i-1)}\;\Delta\mathbf{U} = {}^{t+\Delta t}\mathbf{F}^{ext} - {}^{t+\Delta t}\mathbf{F}^{int(i-1)}$$

$${}^{t+\Delta t}\mathbf{U}^{(i)} = {}^{t+\Delta t}\mathbf{U}^{(i-1)} + \Delta\mathbf{U}^{(i)}$$

Covergence check

if convergence criteria are not satisfied-perform next equlibrium iteration

Tab. 1: Incremental and iterative solution of equilibrium equation in inelastic analysis

In general, we refer to an error in the stress calculation as a consequence of approximations, like for the plastic strains that will be explaining in the text which follows. The algorithm should provide reasonable accuracy for large load increments and the error should rapidly diminish as the load step is decreased. Convergence check is performed through criterium of incremental inner energy.

$$\Delta \mathbf{U}^{(i)\,T}\left({}^{t+\Delta t}\mathbf{F}^{ext} - {}^{t+\Delta t}\mathbf{F}^{int(i-1)}\right) \le \varepsilon_E \Delta \mathbf{U}^{(1)\,T}\left({}^{t+\Delta t}\mathbf{F}^{ext} - {}^{t}\mathbf{F}^{int}\right) \quad (1)$$

where is ε_E - tolerance for energetic convergence criteria.

2 SHELL

Shell element 4-node [4] that is used in analysis is shown on **(Fig. 1)**. Improvement of the response of the shell element is achieved through additional strain corresponding to generalized displacements that are incompatible between elements.

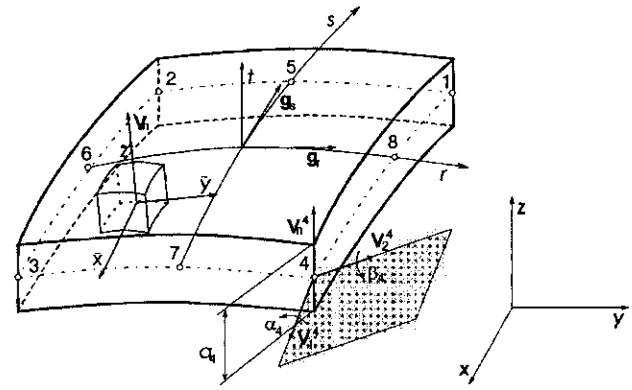

Fig. 1 Shell geometry

Vector of displacement in componential form:

$$\mathbf{u}_i = \sum_{k=1}^{N} \mathbf{h}_k \mathbf{U}_i^k + \frac{\mathbf{t}}{2}\sum_{k=1}^{N} \mathbf{a}_k \mathbf{h}_k (-\mathbf{V}_{2i}^k \alpha_k + \mathbf{V}_{1i}^k \beta_k) \quad (2)$$

where $\mathbf{a}_k$ is shell thickness, $\mathbf{h}_k$ is interpolation function, $\mathbf{U}_i$ is nodal displacement, α_k and β_k are rotation in local coordinate system, $\mathbf{V}_{2i}^k$ and $\mathbf{V}_{1i}^k$ are basic vectors of local coordinate system in node *k*, **(Fig. 1)**.

3 VON MISSES ELASTIC-PLASTIC MATERIAL MODEL WITH MIXED HARDENING

For this material model yield condition [3] is:

$$ {}^{t}\mathbf{f}_y = \frac{1}{2}\,{}^{t}\hat{\mathbf{S}}^T \cdot {}^{t}\hat{\mathbf{S}}' - \frac{1}{3}\,{}^{t}\hat{\sigma}_y^2 = 0 \quad (3)$$

where **S** is deviatoric stress and $\hat{\sigma}_y$ is the yield stress for the uniaxial stress state, **(Fig. 2)**.

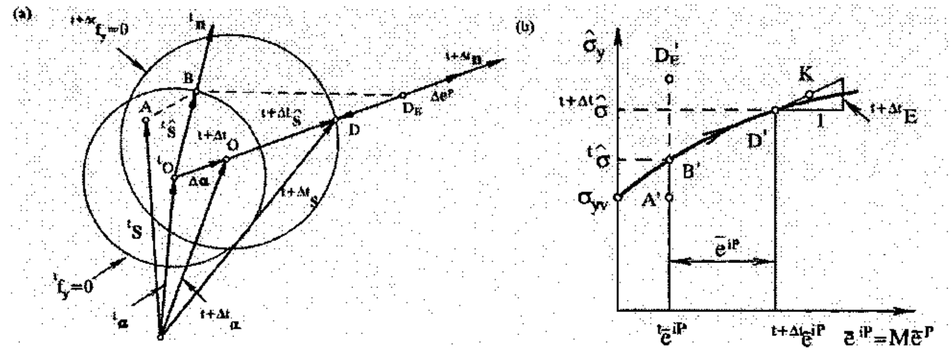

Fig. 2: Stress condition on yield surface on the begining and the end of step

4 APPLICATION OF SOFTWARE CATIA FOR MODELING ROLL BAR

CATIA [5] represent one of the most popular software today for 3D modeling with adaptive tehnology and posibility to modeling real structure. Also this program gives possibility of linear static and dinamic analysis. Pipes of roll bar was modeling like shell.

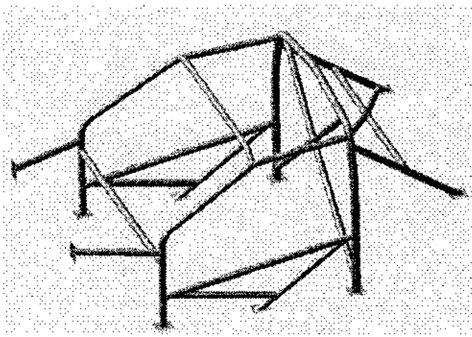

Fig. 3 3D model of roll bar

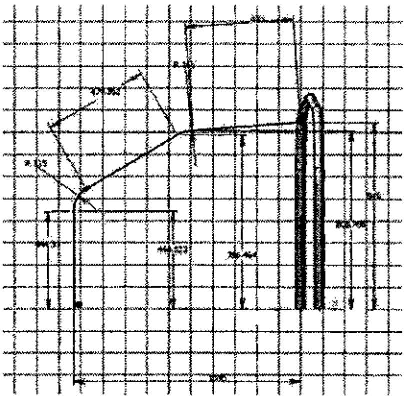

Fig. 4 One step in modeling

5 ANALYSIS OF ROLL BAR IN SOFTWARE PAK

Analysis was done in software PAK [1] and postprosesing was done in software FEMAP [6]. According to FIA standards, roll bar need to contain basic arc, auxiliary arc, front arc, and two lateral arcs, several diagonal arcs and few lamellas. The basic arc that stands behind head of driver represents basic part of structure. In the case of roll over it is exposed to the maximal load and displacement. Mesh is generated in CATIA using izoparametric rectangular shell finite element. After corect montage, it has to prevent deformation on basic arc and to reduce hurting of persons that are in vehicle.

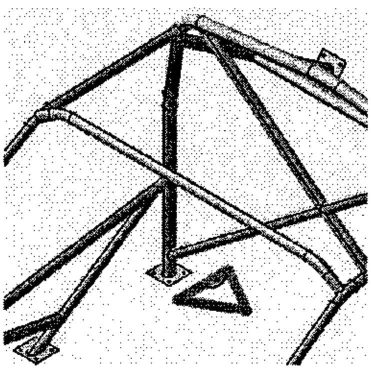

Fig. 5 Way of defyning mesh

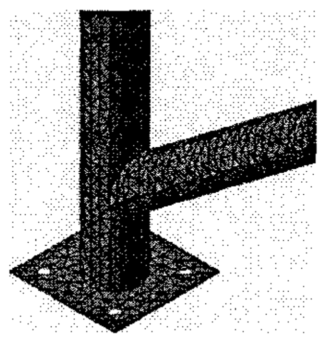

Fig. 6 Finite element on basic arc

Pipe material has to be from steel C1212 or C1220 (ISO standard) with percent of carbon 0.3%. Also they can't have any liquid inside and need to provide free entry in vehicle. According to the standard (FIA) there are different ways of montage, but in any of that case diagonal arc have to be connected in the way not to provoke hurting. Whole structure has to follow shape of car. We used pipe that have diameter 45 mm and 38 mm and thickness 2.5 mm.

Experiments can be done on basic and auxiliary arc. Strength verification, in this case, has to be done on basic arc with loading of 57.7 kN, using rectangular plate 500x200 mm in intervals of 15s. Since cage is observed in whole integrity, test has to be realized on complete structure. Load of 57.7 kN may not provoke flexion more than 50mm and any other damage.

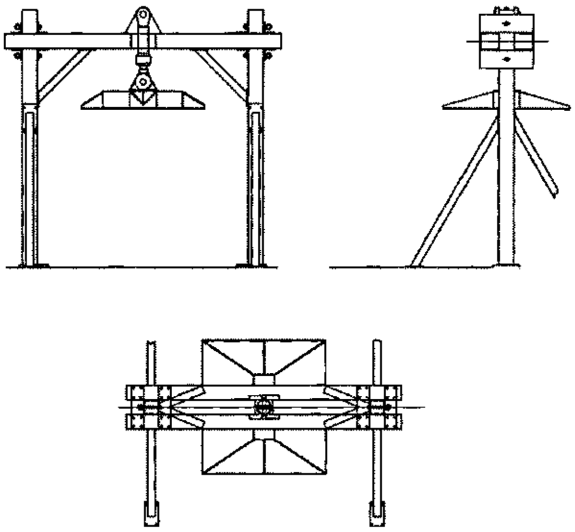

Fig. 7 Device for testing

In numerical analysis we prescribed displacement in the highest point on basic arc of roll bar and for results we got force. Solution is obtained by 50 steps of displacement increments equal to 1mm.
Initially used steel was C1212 that has yield stress 350 MPa. From **Fig. 8** it can be seen that for C1212 we have significant plastic yield before reaching limited load. Because of forming plastic hinge, it will not come to the hardening, which can provoke fracture of structure. Further process of deformation, will going to continue without increasing of external force, which can cause fracture of structure.
Because of that, it was used C1213 that has yield stress 440 MPa. According to the **Fig. 8** we can notice that behavior of structure is almost linear until reaching limited load. For that load displacement is only 10mm (it is five time less than ultimate displacement).
Also it can be noticed that this structure can take higher load than the limited one. Basis on those results it can be conclude that this material satisfied prescribed conditions.

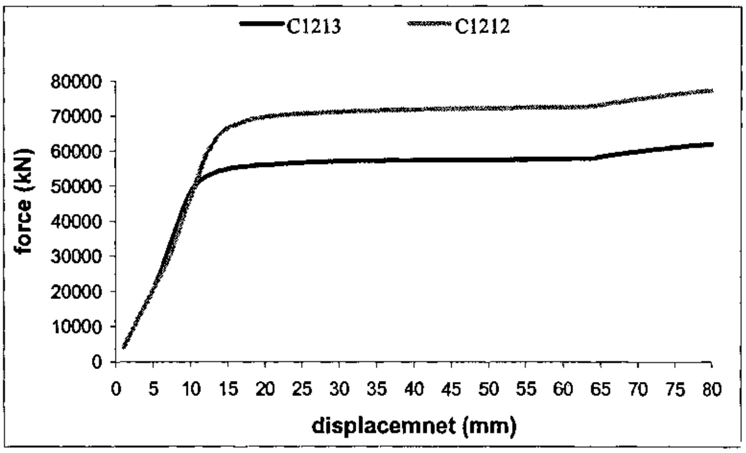

Fig. 8: Force-displacement dependence

The numerical results for field of displacement and field of stress are shown on **Fig. 9** and **Fig. 10**. Maximal plastic strain is in the area of applied load (**Fig. 11**). In the other parts of structure stress is more time less than yield stress (**Fig. 10**).

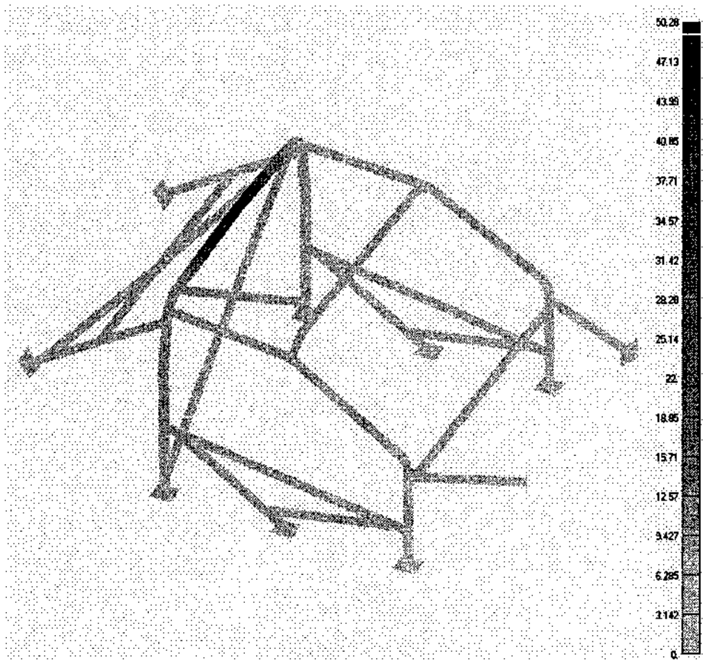

Fig. 9 Field of displacement

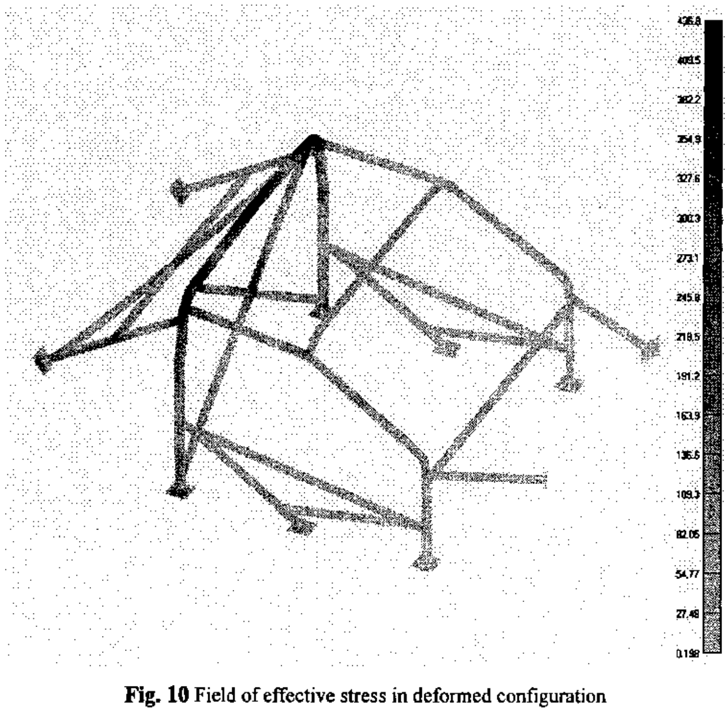

Fig. 10 Field of effective stress in deformed configuration

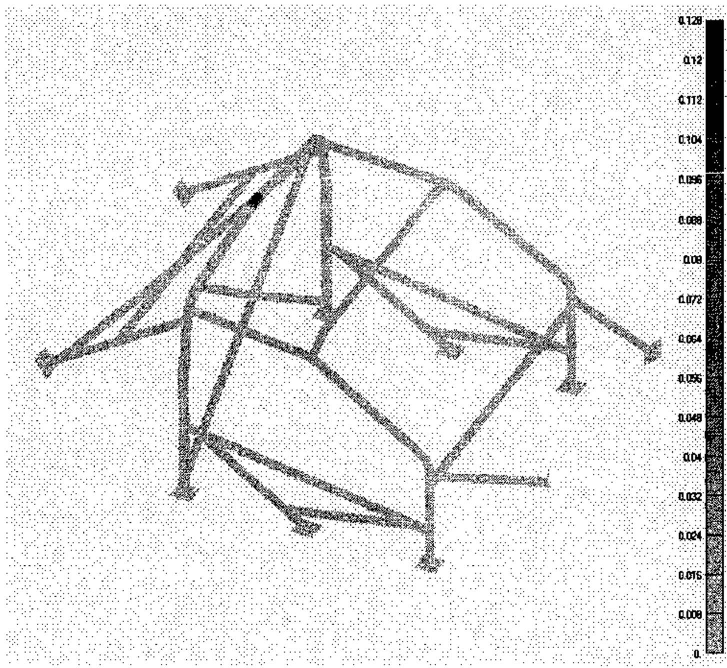

Fig. 11 Field of plastic strain

6 CONCLUSIONS

This analysis approved that material characteristics has significant influence on behavior on this kind of structure. Steel C1212 didn't satisfy prescribed conditions while C1213 did. Using this material structure will resist prescribed conditions (displacement of 50mm and load of 57.7 kN), and we will not have damage. Next step can be optimization of structure so we could put material C1213 on only basic arc of roll bar, and on that way reduced cost.

This kind of modeling and analysis that were performed in short time was reduced coasts and time from creating idea to putting product on market. Procedure that was used in this work also reduced the number of experimental testing and their costs.

7 REFERENCES

[1] KOJIC M., SLAVKOVIC R., ZIVKOVIC M., GRUJOVIC N.: *PAK-S Program for FE Structure Analysis,* Faculty of Mechanical Engineering in Kragujevac, University of Kragujevac, Kragujevac, SCG

[2] KOJIC M., SLAVKOVIC R., ZIVKOVIC M., GRUJOVIC N.: *Finite Element Method - linear analaysis,* Faculty of Mechanical Engineering in Kragujevac, University of Kragujevac, Kragujevac, SCG, 1998

[3] KOJIC M., BATHE K.J.: *Inelastic Analysis of Solids and Structures,* Springer-Verlag, 2005

[4] SLAVKOVIC R., ZIVKOVIC M., KOJIC M.: *Enhanced 8-node three-dimensional solid and 4-node shell elements with incompatible generalized displacement,* Comunication of numerical methods in engineering, Vol. 10, 699-709 (1994)

[5] Software CATIA V5R15

[6] Software FEMAP 8.1