Zeitschrift: IABSE reports = Rapports AIPC = IVBH Berichte
Band: 75 (1996)

Artikel: Three dimensional nonlinear finite element simulation of R/C joints
Autor: Gospodinov, Gospodin / Kinarev, Stefan
DOI: https://doi.org/10.5169/seals-56919

Three Dimensional Nonlinear Finite Element Simulation of R/C Joints

Gospodin GOSPODINOV
Associate Professor
Univ. of Arch., Civil Eng. and Geodesy
Sofia, Bulgaria

Gospodin Gospodinov, born 1946, is an Associate Professor at the Department of Civil Engineering of UACEG, Sofia. His research interests are in numerical methods such as BEM and FEM, fracture mechanics as well as nonlinear numerical modeling and simulation of R/C structures.

Stefan KINAREV
Assistant Professor
Univ. of Arch., Civil Eng. and Geodesy
Sofia, Bulgaria

Stefan Kinarev, born 1949, is an Assistant Professor at the Department of Civil Engineering of UACEG, Sofia. His research interests are in numerical methods, general purpose FE programs and mathematical modeling.

Summary

Three dimensional nonlinear finite element analysis of exterior/interior monolith reinforced concrete joint is presented in this paper. Two types of loading are considered, namely: monotonic and cyclic and five different loading cases are developed. The main sources of nonlinearity, taken into account, are: cracks propagation in the tensile concrete zone, crushing of concrete in compression and hardening plasticity of steel members. The numerical results are compared with experimental data and some conclusions are drawn.

1. Introduction

The results, presented in this paper should be considered as a part of the research, given in the Final Report of COST C1 PECO contract: ERBCIPECT 926033 - see reference [1].
Suppose, we have to perform a static or dynamic analysis of a reinforced concrete frame. The objective is to choose a mathematical model which will enable us to simulate the nonlinear behaviour of the structure, accounting for the real constitutive quantities of the material. In order to accomplish this task properly, the designer usually goes through three main steps: experimental work, local numerical simulation and global numerical (frame) simulation [6], [7]. Since the first step is the most important and expensive, a question is then put forward - what level of the experiment should be carried out? - a point level (simple uniaxial concrete/steel samples), substructure (such as isolated joint, beam , cantilever etc..) or the whole structure itself. Obviously the third alternative, even very attractive and accurate, is not always recommendable because it is too expensive. To get a proper answer of the question posed, we first analyze the diagram of the bending moments in the case of a symmetric frame, loaded laterally as shown in Figure 1. We notice the following:

- there are certain points of zero bending moments;
- it is clear from the Figure 1, that in accordance with the moment diagram, few substructures may be identified, such as : joint, cantilever beam and subframes.

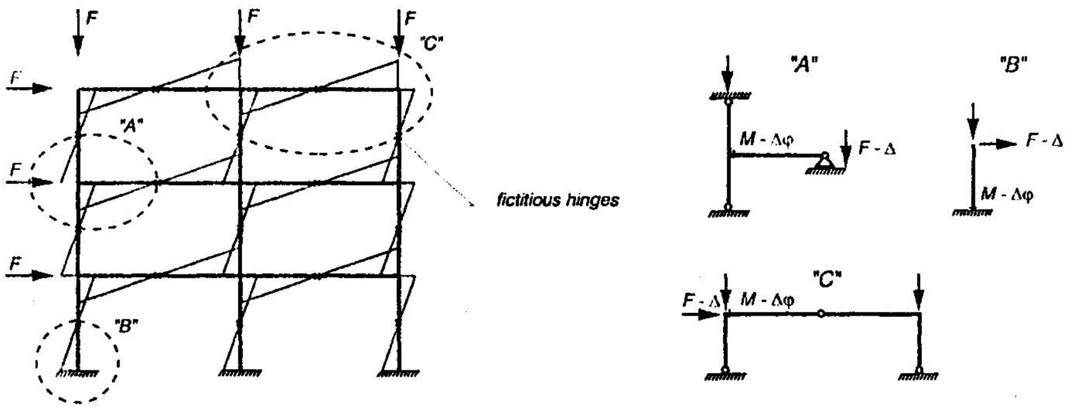

Figure 1. The bending moment diagram of a plane frame, loaded laterally and the identified substractures

It is possible instead of doing experiment on the whole structure or simple uniaxial "point level" experiment (which is cheap, but still unreliable and followed by a tedious FE analysis), to perform experimental and/or numerical work on those substructures, and using these results to create a simplified FE model of the whole structure as an assembly, in order to get the final solution - simple from one hand and reliable from another. As an example, the deformation state of a real R/C joint is shown in Figure 2.a. In Figure 2.b the most simplified possible mechanical model is added. The intention is by using the nonlinear rotational spring to simulate the nonlinear behaviour of the joint. That could be done by adjusting the experimental data to the constitutive parameters of the spring in an integral sense - see references [1], [6] and [7] where an attempt is made for such a numerical simulation.
The experimental work on the substructures can be partly or thoroughly replaced by numerical simulation, provided the mathematical method used and the relevant software tools are reliable enough. Such a refined FE numerical simulation of R/C joints is developed in this paper. The software package used for this purpose is ANSYS51 [2].

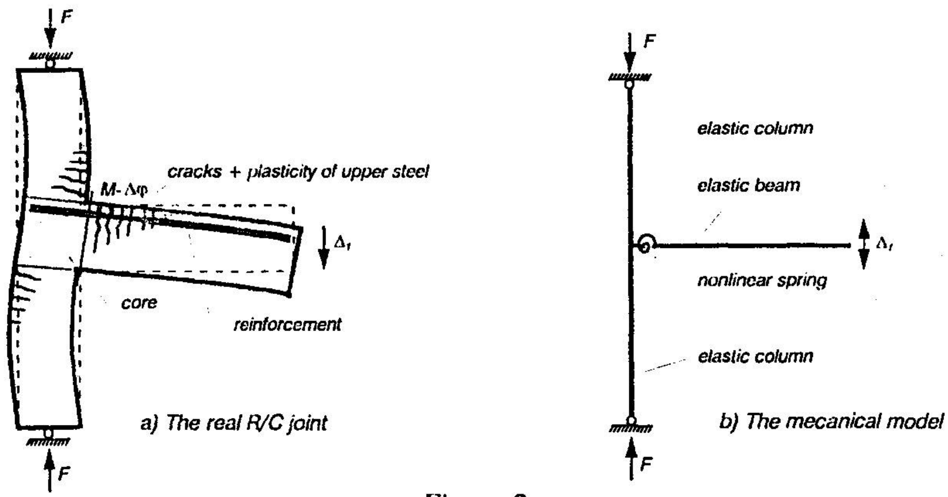

Figure 2

2. The theory

For modelling the concrete behaviour ANSYS51 offers a three - dimensional isoparametric brick finite element with 8 nodes and 24 DOF. The model is capable of cracking in three directions in tension at an integration point. In compression the point may crush when a certain condition is fulfilled The criterion for failure of the concrete due to multiaxial stress state is expressed as follows:

$$\frac{F}{f_c} - S \geq 0, \qquad (1)$$

where F is a function of the principal three dimensional state , S is the failure surface, suggested by William and Warnke (1975) [3],[5], expressed in terms of principal stresses and parameters to be taken from an experiment, f_c is the uniaxial crushing strength.

If equation (1) is not satisfied, there is no cracking or crushing at the concrete point into consideration - the material is considered to be isotropic and linear. When the material fails in triaxial compression, the concrete is assumed to crush and according to the theory adopted, crushing is defined as a complete deterioration of the structural integrity. The material strength is assumed to be zero and there is no contribution to the stiffness from this point. If the failure criterion (1) is satisfied and one of the principal stresses in directions 1, 2 or 3 is positive, cracking occurs in the plane perpendicular to the principal stress. The smeared crack approach is developed, allowing control of the "open" and "close" state of the cracks, so the "material" matrix changes accordingly.

Three-dimensional uniaxial tension-compression spar element with three degrees of freedom is used to represent the steel reinforcement. The spar element assumes a straight bar, axially loaded at its ends, and of uniform properties from end to end. A rate-independent bilinear hardening plasticity is implied, characterized by irreversible straining that occurs in the steel once a certain level of stress is reached. Unloading is assumed to occur elastically.

3. Applications

Based on the above theory and software few numerical examples are developed. A real external reinforced concrete joint, tested by Penelis and el. [4] - see Figure 3, is chosen in order to use the input data and compare the results.

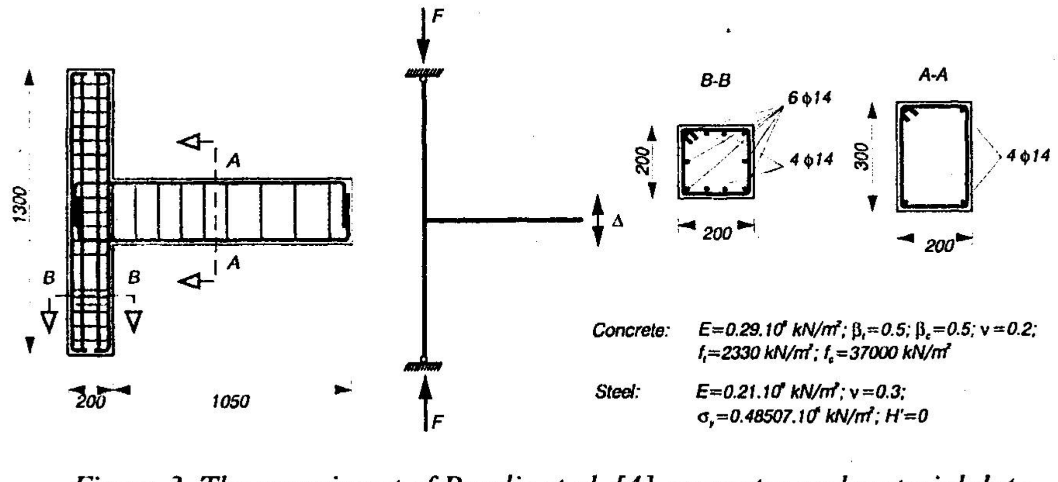

Figure 3. The experiment of Penelis et al. [4], geometry and material data

The loading of the joint is displacement controlled vertical movement of the end of the beam. Two types of R/C joints are numerically analyzed - external and internal. In the first case three loading histories are considered- see Figure 4, and the solution is compared with the experimental results. Two loading histories are developed for the internal R/C joint - see Figure 4, where in x direction the number of loading steps is given in order to read the consequent graphics more easily. The two FE meshes plus some additional data are shown in Figures 5 and 8.

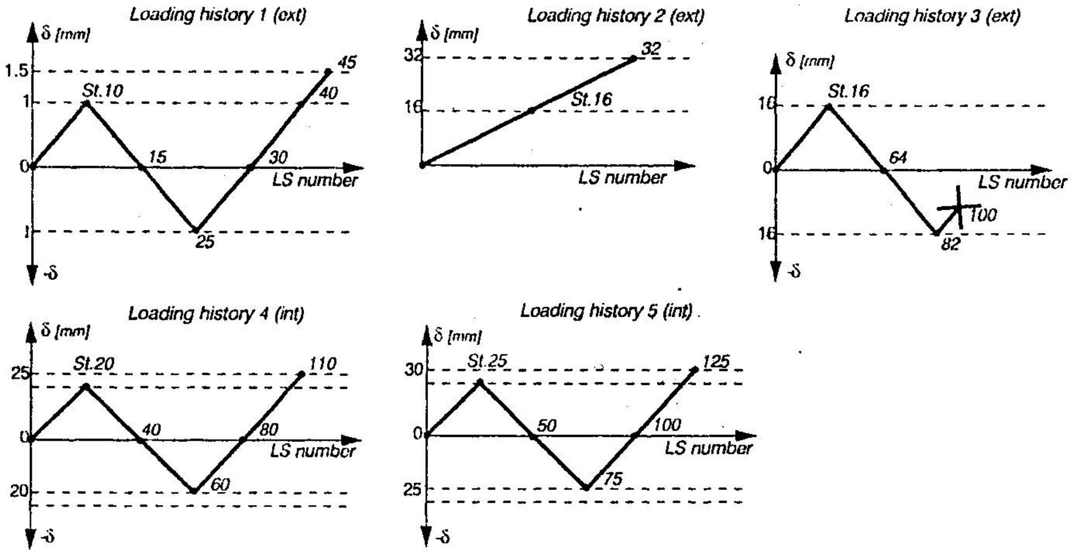

Figure 4. The loading sequence for the loading cases considered

3.1 Loading case 1

The purpose of this numerical test is to investigate the first and very important source of nonlinearity - the crack initialization and propagation in the tensile zone. One and a half hysteretic loops are considered and the maximum displacement is 1.5 mm. The process of damage due to tensile cracks is then monitored and some results at important load steps are given. The propagation and orientation of the cracks at the front of the joint at steps 4 (initial cracking) and 45 (final) are shown in Figure 5. Both - "open" and "closed" status of the cracks at integration points could be accounted, as shown. The essential conclusion is that the model is very successful and reliable as far as the tensile cracking process is concerned. There is no comparison with the experimental results, because they are not available for this loading case.

3.2 Loading case 2

This numerical solution is done for monotonic, displacement controlled vertical movement of the beam end. The displacement is applied incrementally by 1 mm up to 32 mm. At the 7.5 mm, yielding of the steel reinforcement is accounted for. That is clearly seen in Figure 6, where the reaction-displacement relationship is given. It should be mentioned that at the final load step two elements have already crushed, so they do not contribute any more to the stiffness of the joint. Taking into account the good coincidence between experimental and numerical curves, a conclusion can be made, that the model is able to represent properly the monotonic, nonlinear behaviour of R/C joint up to failure load. Three sources of nonlinearity are considered in this numerical test, namely: cracking and crushing of concrete and plasticity of steel reinforcement.

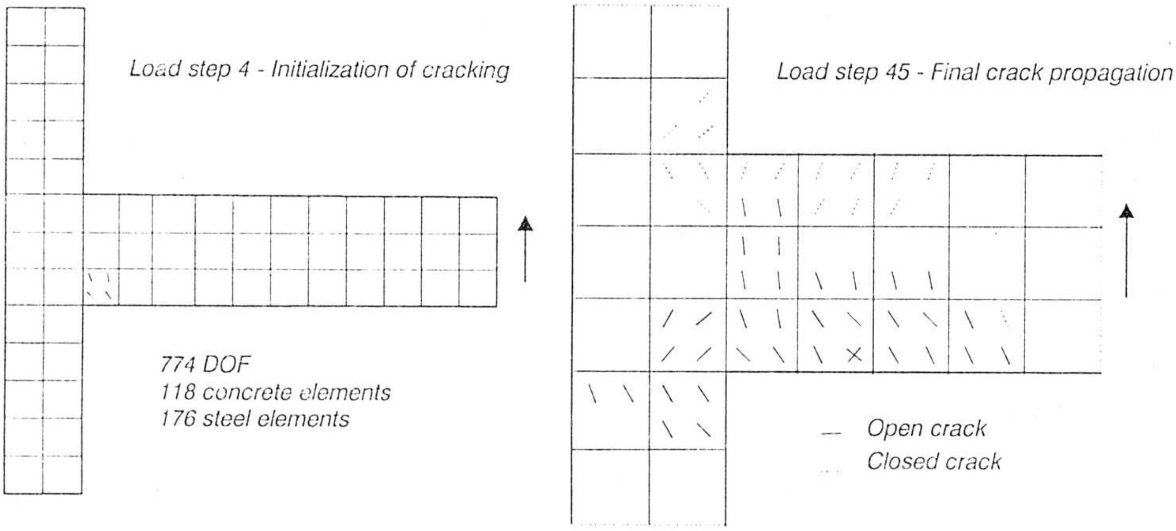

Figure 5. Loading case 1 - FE mesh data and tensile crack development at load step 4 (0.4 mm) and load step 45 (1.5mm)

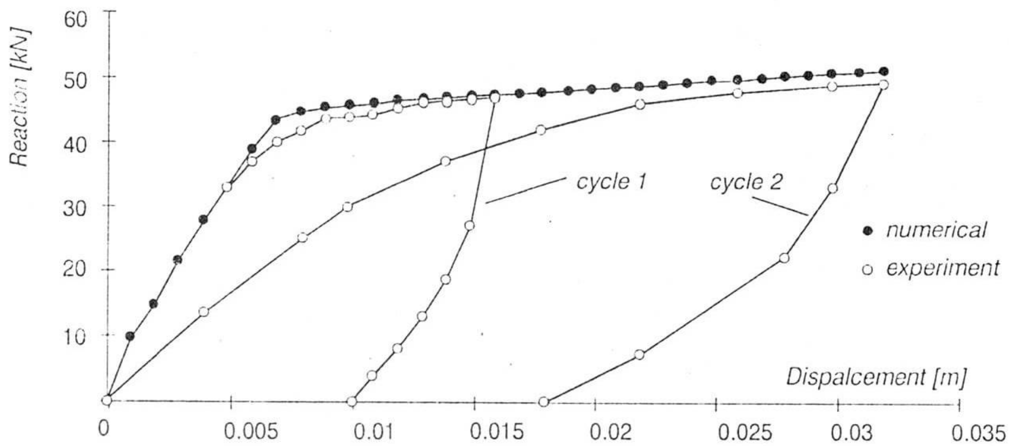

Figure 6. Loading case 2 - Reaction - displacement curve

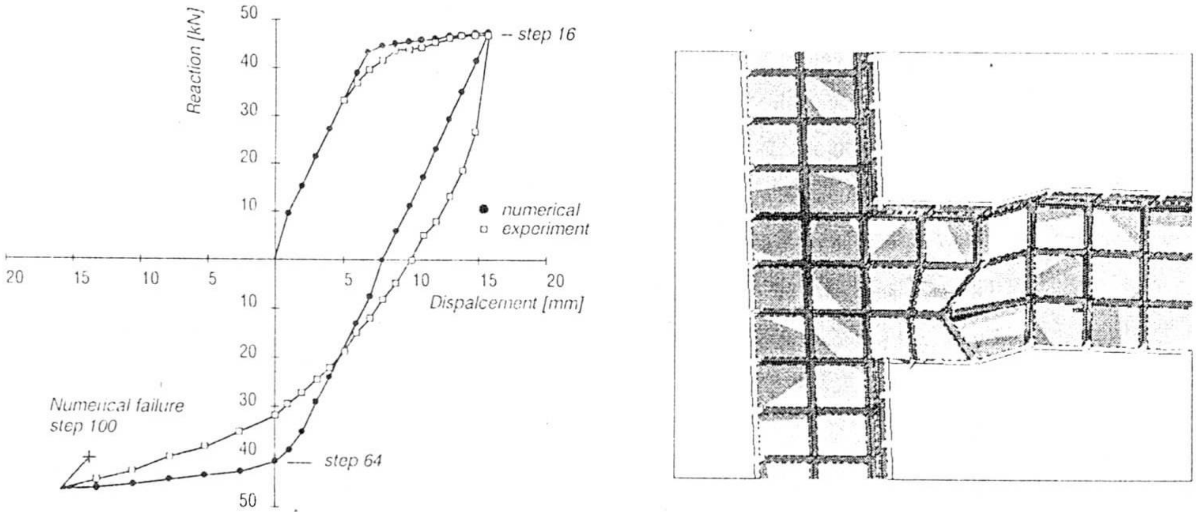

Figure 7. Loading case 3 - Reaction - displacement curve and collapse mechanism

3.3 Loading case 3

Loading case 3 is especially designed, using the same mesh, to develop a hysteretic numerical solution. The positive/negative displacements are just as in the experiment [4]. The displacement during the first cycle is ± 16 mm. The numerical and experimental reaction-displacement curves are plotted in Figure 7. The tensile cracking begins at displacement level 0.4 mm, so it is not indicated clearly on the diagram. At about 7.5 mm the reinforcement begins to yield and the steel plastification of is "on". At step 16 (+16 mm) unloading, according to test data begins and the stiffness is linear, which is typical for the implied theory. There is no good fitting between numerical and experimental curves, particularly when the plasticity of reinforcement is developed. The explanation is that the concrete - steel interaction is not properly simulated. In other words the link of the two materials at nodal points does not allow slipping - which means that the "pull out" effect is not simulated at all. This can be easily overcome if new "link" elements (available in ANSYS) are put into the analysis. After step 64 the numerical solution requires more and more iterations at every new loading step - that is an indication of near failure state of the model. After step 82 a collapse mechanism is formed and failure (nonconvergency) occurs - see the collapse mechanism shown in Figure 7. At the moment of collapse, many elements, situated near the column face are in a "crush" state and a plastic hinge is formed. The column is already free of stresses and as a result the joint can not sustain more loading. Unfortunately, that is not the case with the experiment, so a conclusion should be made that the present modelling can not properly predict the real hysteretic response of the R/C joint. One of the reasons is the small number of concrete elements along the height of the beam. As a result of this poor FE discretization the number of integration points is too small (only 6), therefore the shear capacity of the beam is not sufficient.

3.4 Loading case 4

The loading cases 4 and 5 deal with the simulation of an internal R/C joint. The FE discretization is improved especially on the height of the beam, as indicated above - see Figure 8. The vertical displacement in this case is imposed antisimetrically on the two beams. The vertical displacement is applied incrementally by 1 mm up to ± 20 mm in the two directions. Figure 9 represents the F-δ curve of the joint. As seen from this graphics the most important states of the structural response is well represented, namely: the tension stiffening, the points of yielding of upper and down reinforcement and the consequent "almost perfectly plastic" behaviour of the joint. The first hysteric cycle is finished and the second one is developing as expected. No substantial structural damages are observed.

3.5 Loading case 5

This loading history case is performed on the same internal joint using the same FE mesh. The intention is by increasing the vertical reversal displacements to ± 25 mm to reach the ultimate state, the failure of the joint. The F-δ curve is plotted in Figure 10. The first hysteric loop is finished with no sudden softening of the structure. As the second cycle begins a shear failure of the core is indicated at load step 104 - at plus 4 mm displacement. The joint softens in almost brittle manner but it is still capable to resist. The deformed shape of the core is shown in Figure 11. At 18 mm displacement, a shear failure of the columns happens in a brittle manner and a big drop of the reaction is indicated. The shear collapse mechanism is shown in Figure 12 and this loading point is accepted as a failure point of the joint.

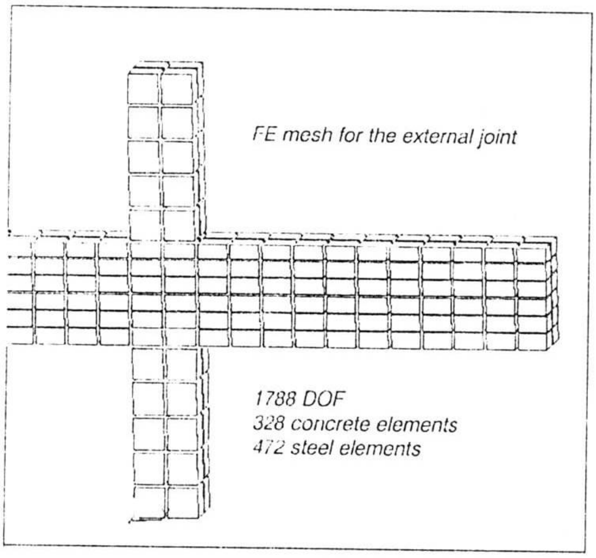

Figure 8. Internal joint - FEM data

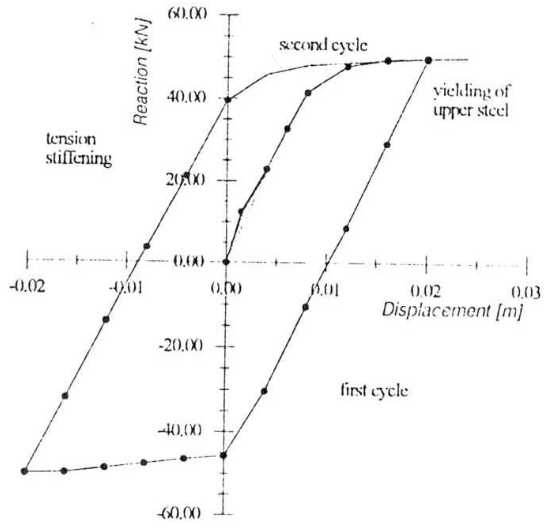

Figure 9. Loading case 4 - Reaction-displ. curve

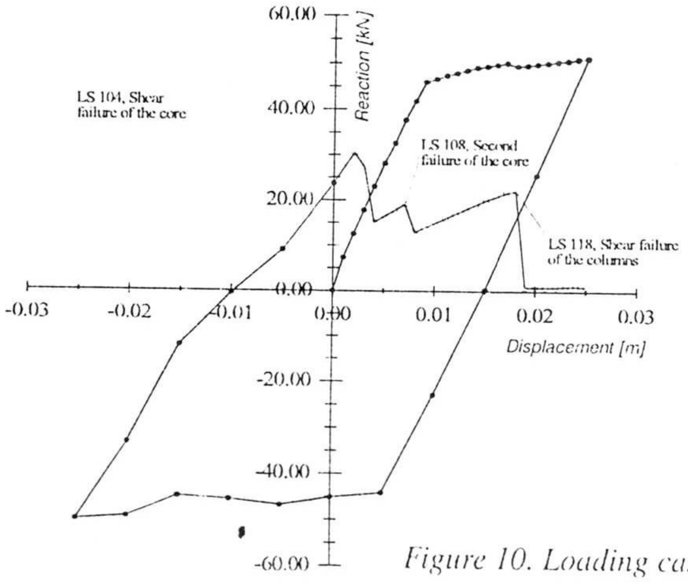

Figure 10. Loading case 5 - Reaction-displ. curve

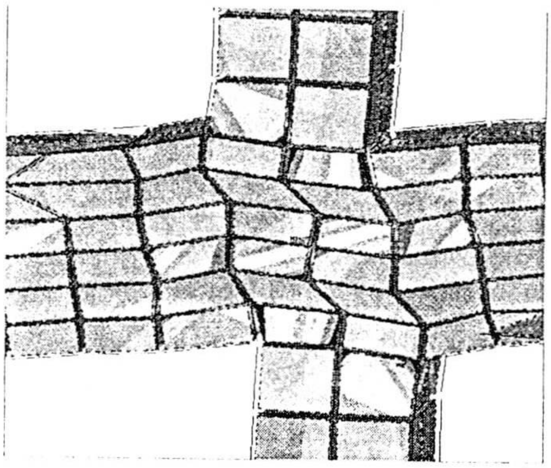

Figure 11. Shear failure of the core

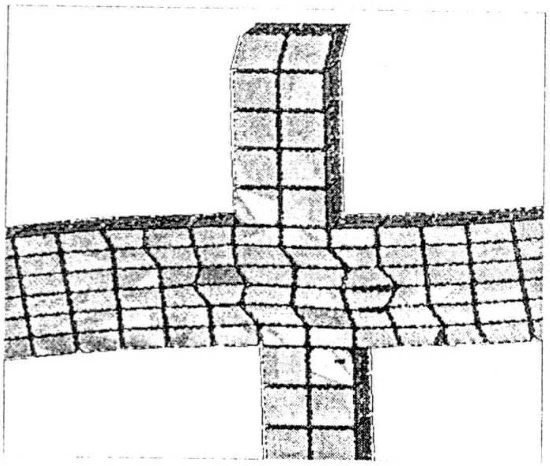

Figure 12. Final shear failure of the columns

4. Conclusions

On the basis of the above analysis the following conclusions can be summarized:

1. The used William-Warnke criterion and the pressure dependent failure surface itself, represent well this type of mathematical simulation.
2. The model indicates a good performance in the case of tensile cracking - the opening and closing processes are reproduced well.
3. Two types of fracturing can be successfully handled, namely: "cracking" type and "crushing" type at low pressure zone. When a big uniform pressure is available and the damage is big, which is very typical for cyclic loading and when the structure is near to ultimate state, a refined FE mesh is needed in order to prevent the numerical failure due to low number of integration points along the height of the beam.
4. The present modelling of reinforcement is working satisfactorily.
5. The "unloading" and "reloading" path are identical and no damage effects take place when repeated loads occur. The energy dissipation is not simulated properly, which is a drawback of the theory implemented.

The important conclusion is that the present three dimensional modelling of R/C joints can be successfully applied in the case of monotonic loading. It allows to reproduce the main features of the joint behaviour observed experimentally and, with a careful selection some important constitutive parameters of the model can be obtained for further use in the nonlinear R/C frame simulation [7]. When a cyclic loading is considered the application of such a theory requires additional research and improvement.

5. References

[1] **Gospodinov G, Vassilev T. , Petkov Z. and Kinarev S.**, "Numerical Investigation of Reinforced Concrete Joints", Final Report of COST C1 PECO contract ERBCIPECT 926033, 1995.

[2] **ANSYS** - Theory and Elements Manuals.

[3] **Chen W.F.**, "Plasticity in Reinforced Concrete", McGraw-Hill, 1982.

[4] **Penelis G., et al.**, "Seismic Resistance of Type 2 Exterior Beam-Column Joints Reinforced with Inclined Bars", ACI Str. Journal, 1-2 1992.

[5] **Beshara F. and Virdi K.**, "Non-linear Finite Element Dynamic Analysis of Two-dimensional Concrete Structures", Proc. of 4-th Int. Conf. on Civil and Str. Eng. Computing, vol.1, Ed. B.H.V. Topping, Civil-Comp Press, 1989.

[6] **Vassilev T. Gospodinov G., and Petkov Z.**, "Numerical Modeling of Hysteretic Behaviour of R/C Joints", Proc. of 2-nd COST C1 workshop on Semi-Rigid Behaviour, Prague, 1994.

[7] **Petkov Z., , Vassilev T. and Gospodinov G.**, "Numerical Analysis of Plane R/C Frames Using Joint Test Data", Proc. of 2-nd COST C1 workshop on Semi-Rigid Behaviour, Prague, 1994.

Acknowledgment: *The authors would like to express their gratitude to professor K. S. Virdi, WG6 COST C1 Chairman, for his support and useful discussions during the course of this work.*