Zeitschrift: IABSE reports = Rapports AIPC = IVBH Berichte

Band: 75 (1996)

Rubrik: Session 6: Modelling of connection behaviour 2

SESSION 6

MODELLING OF CONNECTION BEHAVIOUR 2

Leere Seite
Blank page
Page vide

The required rotation capacity of joints in braced steel frames

Etienne BOENDER
Structural Engineer
University of Technology
Delft

Fluor Daniel B.V.
Haarlem

The Netherlands

Jan STARK
Professor
University of Technology
Delft

TNO Building and Construction Research
Delft
The Netherlands

Martin STEENHUIS
Research Assistant
TNO Building and Construction Research
Delft

The Netherlands

Summary

One of the conditions for the use of plastic design for braced frames is that the joints have sufficient rotation capacity. In current practice, the requirements for rotation capacity are determined with the beam line theory. For a beam in an inner bay, this is reasonable; for a beam in an outer bay, this is questionable because of the deformations of the columns and difference in the joints.

In this paper an analytical model is presented for the determination of the required rotation capacity of joints. This model includes parameters like the resistance and stiffness of beams, columns and joints, as well as the second order effects in the columns.

The analytical model, however, is complicated for use in practice. Therefore, the beam line theory has been compared to this model, in order to investigate whether the beam line theory can be used safely for side spans. The results of a parameter study has shown that in certain cases, the beam line theory predicts too small (unsafe) values for the required rotation capacity compared to the analytical model. However, if the resistance of the joint connecting the beam to the outer column is smaller than half of the beam resistance, the ultimate load will not be reduced by more than 5% when the beam line theory is used. In other cases, a modification factor should be applied to the results of the beam line theory.

List of symbols

EI_{cln} is the stiffness of a column;
EI_{bm} is the stiffness of a beam;
f_{mod} is a modification factor;
F_E is the Euler buckling load;
k_m is a calculation factor;
l_{bm} is the length of a beam;
$M_{Rd.bm}$ is the plastic moment capacity of a beam;

$M_{Rd.cln}$ is the plastic moment capacity of a column;
$M_{Rd.i}$ is the moment capacity of a joint i;
$M_{Rd.m}$ is the moment capacity of a joint to an inner column (mid joint);
$M_{Rd.s}$ is the smaller of the moment capacity of a joint to an outer column (side joint) and the resistance of an outer column for a moment transferred from the connected beam to the column;
N_{Sd} is the axial force in a column;
P is the axial force acting at an outer column;
q is the uniformly distributed load on a beam;
$S_{j.m}$ is the stiffness of a joint to an inner column;
$S_{j.s}$ is the stiffness of a joint to an outer column;
S_s is the stiffness of a spring representing the behaviour of a joint to an outer column and this outer column;
$\alpha_1 l_{bm}$ is the length of a part of a column under a beam;
$\alpha_2 l_{bm}$ is the length of a part of a column above a beam;
β is the relative stiffness;
ζ is a calculation factor;
ν is a multiplication factor for second order effects;
ρ_m is the relative stiffness for a joint to an inner column;
ρ_s is the relative stiffness for a joint to an outer column;
$\phi_{j.i}$ is the required rotation capacity of a joint i;
ϕ_s is the required rotation capacity of a spring representing a joint to an outer column and this outer column;
$\phi_{j.m}$ is the required rotation capacity of a joint to an inner column;
$\phi_{j.s}$ is the required rotation capacity of a joint to an outer column.

1. Introduction

The response of steel frames is influenced by the resistance, stiffness and rotation capacity of the joints. Eurocode 3 Annex J [1] provides rules for the determination of the resistance and stiffness. For backgrounds to Eurocode 3 Annex J refer to [2, 3]. For the rotation capacity, only a set of "deemed to satisfy" rules is given. For example, it is stated that failure of a column web in shear will lead to sufficient rotation capacity. Joints not complying with these rules do not necessarily have insufficient rotational capacity in all cases. Then the verification should be based on a concept in which the available rotation capacity of a joint is compared with the required rotation capacity as determined in the frame analysis.

This paper reports the results of a study on the required rotation capacity in braced steel frames [4]. A review on required rotation capacity in braced frames was published by Bijlaard [5]. In the paper of Bijlaard the determination of the rotation capacity of joints was based on the beam line theory. The beam line theory assumes that the columns in the frame remain straight. This is of course questionable for outer columns, because their deformations (due to bending moments and second order effects) do have an influence on the response of the frame and thus the rotations in the joints.

This paper gives in section 2 a recollection of the beam line theory. In section 3, an analytical model for the prediction of the required rotation capacity is presented. This analytical model takes into account the deformations of the outer column. In section 4, the analytical model is simplified by neglecting the influence of the column. It appears that in certain cases, predictions with the simple model are on the unsafe side. In section 5 the situations are determined where this unsafety is significant. In the last section, modification of the simple model is proposed.

2. Beam line theory

The beam line theory for the determination of rotation capacity in braced frames is based on the assumptions that:
- the columns remain straight;
- the two joints at both ends of the beam are identical;
- plastic hinges form in the joints.

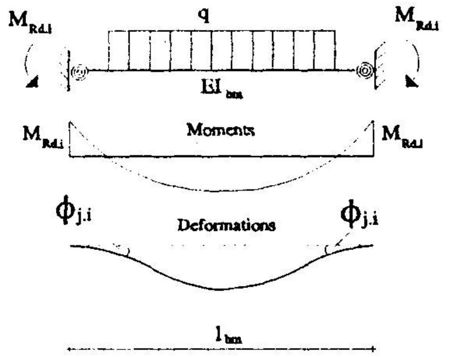


Fig.1. Scheme for beam line

The system of the beam between two rotational springs is shown in fig. 1. The beam is loaded with a uniformly distributed load q. The required rotation capacity of the joints is:

$$\phi_{j.i} = \frac{q l_{bm}^3}{24EI_{bm}} - \frac{M_{Rd.i} l_{bm}}{2EI_{bm}} \tag{1}$$

3. The analytical model

The analytical model [4] is based on elastic-rigid plastic frame theory. This theory allows analytical treatment of the problem and represents the actual frame behaviour in a sufficient accurate way. The relation between the moment and the rotation of the joint is assumed to be bi-linear, see fig. 2. In this figure the angle S_j represents the stiffness of the joint, M_{Rd} the strength and ϕ_{max} the maximum rotation. The moment curvature relation of beams and columns are also assumed to be bi-linear, see fig. 3.

A braced steel frame generally consists of storeys and bays. The beams in the outside bays are called side beams; the other beams are internal beams. For the analysis of the beams and joints, a sub frame is considered. This sub frame consists of a side beam and a part of an outer column: the length of the column is taken as two times half the storey height, see fig. 4. When the sub frame is at the first storey level, the length of the column below the beam is equal to the storey height. The beam is loaded with a uniformly distributed load q and the column with an axial force P.

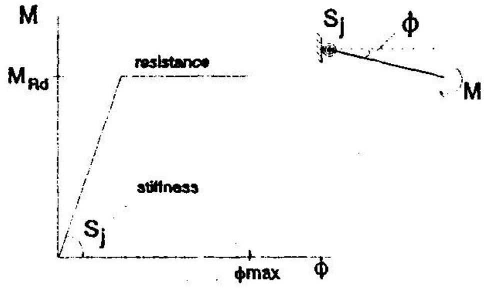


Fig. 2. M-ϕ diagram of a joint

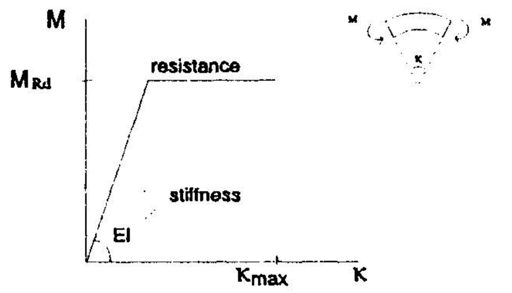


Fig. 3. M-kappa diagram of a member

This results in the scheme of the analytical model as given in fig. 5. The joint at the outer column is referred to as the side joint; the joint at the inner column as the mid joint.

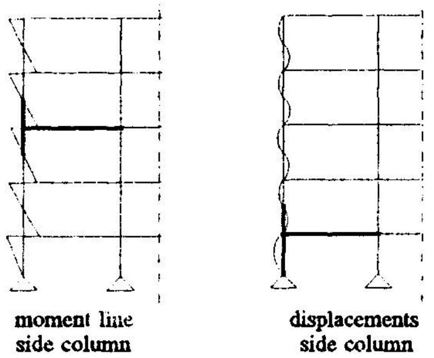


Fig. 4. Moments and displacements of the outer column

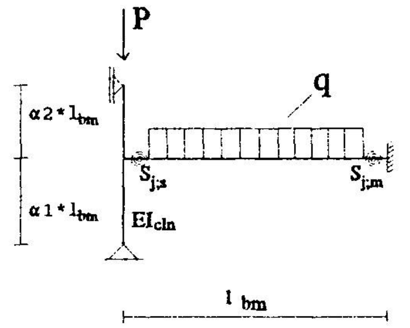


Fig. 5. Scheme of the analytical model

3.1 Beam behaviour

The behaviour of the beam in the sub frame can be analyzed with a modified beam line model as given in fig. 6. The side spring (on the left beam end) represents the behaviour of the outer column and the joint to this column in terms of resistance $M_{Rd.s}$ and stiffness S_s. The spring on the right beam end represents the behaviour of the joint to the inner column (mid joint) in terms of resistance $M_{Rd.m}$ and stiffness $S_{j.m}$.

The maximum uniformly distributed load $q = 8\ (M_{pl.bm} + 0{,}5\ M_{Rd.m} + 0{,}5\ M_{Rd.s})\ /\ l_{bm}^2$ is reached when a full mechanism develops. A full mechanism develops when 'the last (third) plastic hinge' forms. At this stage the maximum rotation is reached in the springs. There are three different locations where a plastic hinge can form: in the two springs and in the span of the beam. So there are three possible 'end situations': the last plastic hinge forms in the side spring, in the mid joint or in the span of the beam. If a plastic hinge forms in the span of the beam, it is assumed that it forms exactly in the mid of the span. This gives only small errors in the prediction of the required rotational capacity of the joints, see [4].

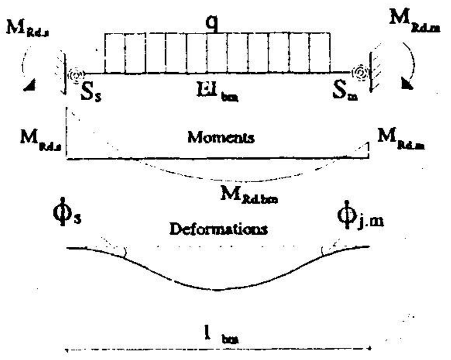
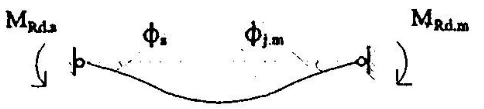


Fig.6. Scheme for a beam in sub frame

The 'end situation' can be determined as follows. Assume that the last plastic hinge forms at mid span as shown in fig. 7.



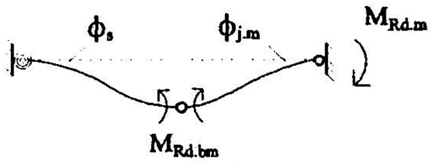


Fig. 7. Last hinge in span beam *Fig. 8. Last hinge in side spring*

In that case, the rotations of the beam ends will exceed the elastic rotation of the side spring respectively the elastic rotation of the mid joint:

$$\phi_s \geq \frac{M_{Rd.s}}{S_s} \quad \text{and} \quad \phi_{j.m} \geq \frac{M_{Rd.m}}{S_m} \qquad (2)$$

In analogy to equation 1, for the rotations in the beam near the joints can be written:

$$\phi_s = \frac{q l_{bm}^3}{24 EI_{bm}} - \frac{M_{Rd.s} l_{bm}}{3 EI_{bm}} - \frac{M_{Rd.m} l_{bm}}{6 EI_{bm}} = \frac{M_{Rd.bm} l_{bm}}{3 EI_{bm}} - \frac{M_{Rd.s} l_{bm}}{6 EI_{bm}} \qquad (3)$$

$$\phi_{j.m} = \frac{q l_{bm}^3}{24 EI_{bm}} - \frac{M_{Rd.m} l_{bm}}{3 EI_{bm}} - \frac{M_{Rd.s} l_{bm}}{6 EI_{bm}} = \frac{M_{Rd.bm} l_{bm}}{3 EI_{bm}} - \frac{M_{Rd.m} l_{bm}}{6 EI_{bm}} \qquad (4)$$

If the last plastic hinge does not form in the span of the beam, then we assume that the last plastic hinge forms in the side spring as shown in fig. 8. In that case for the mid joint should hold: $\phi_{j.m} \geq M_{Rd.m} / S_{j.m}$. The rotation of the mid joint is:

$$\phi_{j.m} = \frac{M_{Rd.s}}{S_s} - \frac{(M_{Rd.s} - M_{Rd.m}) l_{bm}}{6 EI_{bm}} \qquad (5)$$

If the last plastic hinge doesn't form in the side spring, it can be concluded that it will form in the mid joint, as shown in fig. 9.

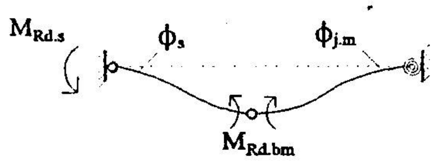


Fig. 9. Last hinge in mid joint

In that case, the rotation of the side spring is:

$$\phi_s = \frac{M_{Rd.m}}{S_{j.m}} - \frac{(M_{Rd.m} - M_{Rd.s})l_{bm}}{6EI_{bm}} \tag{6}$$

3.2 Behaviour of the outer column and the side joint

In paragraph 3.1 it was assumed that the side spring is bi-linear. In reality, the side spring should represent the behaviour of the outer column and the behaviour of the joint connecting the beam to the outer column.

The behaviour of the outer column is complex because of second order effects and the fact that one or two plastic hinges may form in this column. These hinges may form due to moments transferred from the beam to the outer column.

The rotation of side spring ϕ_s is constituted from rotation of the column ϕ_{cln} and the rotation in the side joint $\phi_{j.s}$. Conservatively, it is assumed that the rotation of the outer column is based on elastic behaviour ($\phi_{cln} = M_{Rd.s} / S_{cln}$). In other words, if plasticy occurs in the column, the corresponding plastic rotations will be assigned to the side joint, so:

$$\phi_{j.s} = \phi_s - \frac{M_{Rd.s}}{S_s} \tag{7}$$

The moment capacity $M_{Rd.s}$ of the side spring is the smaller of:

- the resistance of the outer column for bending moments transferred from the beam to the outer column and
- the resistance of the side joint.

The stiffness of the side spring is influenced by the stiffness of the side joint and the outer column as follows:

$$\frac{1}{S_s} = \frac{1}{S_{j.s}} + \frac{1}{S_{cln}} \tag{8}$$

For the stiffness of an outer column with pinned base can be written:

$$S_{cln} = \frac{\alpha_1 + \alpha_2}{\alpha_1 \alpha_2} \frac{3EI_{cln}}{l_{bm}} \tag{9}$$

The second order effects of the column can be been taken into account by the multiplication factor n/(n-1), n = F_E / N_{Sd}. The rotation in the column at floor level is then:

$$(\phi_{cln})^{\text{second order}} = \frac{n}{n-1} (\phi_{cln})^{\text{first order}} \tag{10}$$

Equations (9) and (10) result in:

$$\frac{1}{S_{cln}} = \frac{n}{n-1} \frac{\alpha_1 \alpha_2}{3(\alpha_1 + \alpha_2)} \frac{l_{bm}}{EI_{cln}} \tag{11}$$

3.3 Behaviour of the sub frame

Equations (2) to (11) give a complete description of the behaviour of the joints in the sub frame. The presentation of these equations can be improved by introducing:

$$\gamma = \frac{\alpha_1 \alpha_2}{3(\alpha_1 + \alpha_2)} \quad \text{for an outer column with pinned base} \tag{12}$$

$$\gamma = \frac{\alpha_1 \alpha_2}{4\alpha_2 + 3\alpha_1} \quad \text{for an outer column with rigid base} \tag{13}$$

$$\rho_s = \frac{S_{j.s} l_{bm}}{EI_{bm}} \quad ; \quad \rho_m = \frac{S_{j.m} l_{bm}}{EI_{bm}} \tag{14}$$

$$\nu = \frac{n}{n-1} \quad ; \quad n = \frac{F_E}{N_{sd}} \quad ; \quad \beta = \frac{EI_{cln}}{EI_{bm}} \tag{15}$$

To determine where the last hinge forms, fig. 10 can be used. The rotations of the joints can be calculated by rewriting formulae (2) to (11). When the last plastic hinge forms in the span of the beam the rotations are as follows:

$$\phi_{j.s} = \frac{(2\beta M_{Rd.bm} - \beta M_{Rd.s} - 6\nu\gamma M_{Rd.s}) l_{bm}}{6\beta EI_{bm}} \tag{16}$$

$$\phi_{j.m} = \frac{(2M_{Rd.bm} - M_{Rd.m}) l_{bm}}{6EI_{bm}} \tag{17}$$

When the last plastic hinge forms in the side joint, the rotations are:

$$\phi_{j.s} = \frac{M_{Rd.s}}{S_{j.s}} \quad ; \quad \phi_{j.m} = \frac{6(\beta + \nu\gamma\rho_s) l_{bm} M_{Rd.s} + \rho_s \beta (M_{Rd.s} - M_{Rd.m}) l_{bm}}{6\beta \rho_s EI_{bm}} \tag{18}$$

When the last plastic hinge forms in the mid joint, the rotations are:

$$\phi_{j.s} = \frac{6\beta M_{Rd.m} l + \rho_m \beta (M_{Rd.m} - M_{Rd.s}) l_{bm} - 6\rho_m \nu\gamma M_{Rd.s} l_{bm}}{6\rho_m \beta EI_{bm}} \quad ; \quad \phi_{j.m} = \frac{M_{Rd.m}}{S_{j.m}} \tag{19}$$

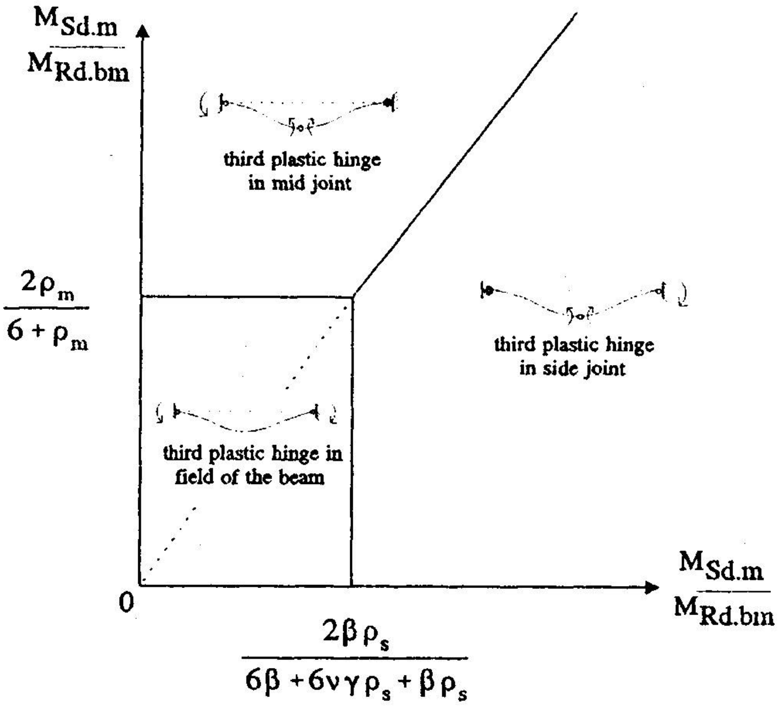


Fig. 10. Development of plastic hinges in the sub frame

4. Simplification

With the analytical model accurate values of the required rotation capacity can be determined. However, the model is complicated for use by practitioners. Therefore, simplification is necessary. To simplify the analytical model, the outer column's influence is removed from this model by assuming that this column remains straight. This results in a simple model, i.e. a beam between two straight columns, with two different springs in strength and stiffness. The simple model is identical to the model of figure 6, but $S_s = S_{j.s}$ and $\phi_s = \phi_{j.s}$.

With the simple model, the required rotation capacity can be predicted as follows. When equation (20) is fulfilled, then the last plastic hinge forms in the mid span of the beam. In that case, the rotation capacities of the side joint $\phi_{j.s}$ and the mid joint $\phi_{j.m}$ can be determined with equations (3) and (4). Otherwise, when equation (21) is fulfilled, then the last plastic hinge forms in the side joint. The rotation capacity of the mid joint $\phi_{j.m}$ can be determined with equation (5). When the last hinge forms in the mid joint, the required rotation capacity for the side joint $\phi_{j.s}$ can be determined with equation (6).

$$\frac{M_{Rd.s}}{M_{Rd.bm}} \leq \frac{2\rho_s}{6+\rho_s} \quad \wedge \quad \frac{M_{Rd.m}}{M_{Rd.bm}} \leq \frac{2\rho_m}{6+\rho_m} \tag{20}$$

$$\frac{M_{Rd.m}}{M_{Rd.s}} \leq \frac{\rho_m(6+\rho_s)}{\rho_s(6+\rho_m)} \tag{21}$$

5. Comparison of the two models

The two models described sections 3 and 4 have been compared. The simple model ideally should predict the required rotation capacity as calculated with the analytical model, or a higher value of this rotation capacity. Actually the analytical model is the same as the simple model, only the outer column has more flexibility. In the analytical model the column takes part in the rotation of the beam's end. In that case, the required rotation capacity of the side joint is smaller than in the case of a straight column (the simple model). Thus the simple model always predicts too large rotations for the side joint.

For the mid joint holds: the stronger the side joint and the more flexible the outer column is, the later the last plastic hinge forms in the side joint. In this case, the simple model, which already would have formed the last plastic hinge, calculates too small rotations $\phi_{j.m}$.

When the rotation capacity in the mid joint $\phi_{j.m}$, as determined by the simple model, is smaller than the rotation $\phi_{j.m}$ found with equation (18), the plastic mechanism in the beam will not be fully reached and the uniformly distributed load q will be lower than 8 ($M_{Rd.bm}$ + 0,5 $M_{Rd.s}$ + 0,5 $M_{Rd.m}$) / l_{bm}^2. In that case, q at failure can be expressed as a function of $\phi_{j.m}$:

$$q_{max} = \frac{24k_m EI}{(6+k_m)l_{bm}^2} * \phi_{j.m} + \frac{24(2M_{Rd.bm} + M_{Rd.m} + 8k_m(M_{Rd.bm} + M_{Rd.m})}{(6+k_m)l_{bm}^2} \qquad (22)$$

with:

$$k_m = \frac{\beta \rho_m}{\beta + \nu \gamma \rho_s} \qquad (23)$$

In equation (22), the rotation in the mid joint $\phi_{j.m}$ should be determined by the simple model.

It is assumed that a reduction of q (error on the unsafe side) of 5% is acceptable. By means of a parameter study, the situations were determined when the error exceeded 5%.

6. Modification factor

From the parameter study it appeared that the reduction of q is never exceeding 5% when the strength of the side joint is smaller than 0.5 $M_{pl.bm}$. When the strength of a side joint is more than 0.5 $M_{pl.bm}$, the required rotation capacity, as found with the simple model, has to be multiplied with a modification factor. For the determination of this modification factor it is referred to [4]. This modification factor is based on upper bounds of the second order factor ν and the geometry factor γ_r and is equal to:

$$f_{mod} = \left(\frac{6}{\rho_s} + \frac{1}{\beta} + 1\right)\frac{M_{Rd.s}}{M_{Rd.bm}} - 1 \geq 1 \qquad (24)$$

The parameter study also showed that in the majority of cases the last plastic hinge forms in the span of the beam. The reduction of q has also been investigated, by assuming that in the simple model the last hinge always forms in the beam span. Further, the reduction factor has been applied in case the strength of the side joint is more than half the beam resistance. It proved for the frames investigated, that the reduction of q is never more than 5%.

7. Conclusions

By adopting elastic-rigid plastic frame behaviour, the required rotation capacity of joints in the outer bays of braced steel frames can be determined analytically. To make this analytical model suitable for use in practice, it can be simplified to a 'modified beam line model'. In this model is assumed that the outer columns remain straight. The value of the rotation capacity according to the simple model has to be multiplied with a modification factor in case the resistance of the joint to the outer column exceeds half of the beam resistance. This 'modified beam line model' can be written as:

$$\phi_{j.s} = \frac{(2M_{Rd.bm} - M_{Rd.s})l_{bm}}{6EI_{bm}} \quad ; \quad \phi_{j.m} = \frac{(2M_{Rd.bm} - M_{Rd.m})l_{bm}}{6EI_{bm}} \cdot f_{mod}$$

In case $M_{Rd.s} \leq 0.5 \cdot M_{Rd.bm}$, $f_{mod} = 1$, otherwise:

$$f_{mod} = \left\{ \frac{6EI_{bm}}{S_{j.s} l_{bm}} + \frac{EI_{bm}}{EI_{cln}} + 1 \right\} \cdot \frac{M_{Rd.s}}{M_{Rd.bm}} - 1 \geq 1$$

References

[1] EUROCODE 3, ENV - 1993-1-1, *Revised annex J*, Design of Steel Structures, CEN, European Committee for Standardization, Document CEN / TC 250 / SC 3 - N 419 E, Brussels, Belgium, June 1994.

[2] Zoetemeijer P., *Summary of the research to bolted end plate connections*, Report 6-85-7, University of Technology, Delft,the Netherlands, 1985.

[3] Jaspart, J.P., *Etude de la semi-rigidite des noeuds poutre-colonne et son enfluence sur la resistance et la stabilite des ossatures en acier*, University of Liège, Belgium, 1991.

[4] Boender E.H., *Study to required rotation capacity of joints in braced steel frames (in Dutch)*, report 95-CON-R0879, University of Technology and TNO Building and Construction Research, Delft, the Netherlands, 1995

[5] Bijlaard F.S.K., *Requirements for Welded and Bolted Beam-to-Column Connections in Non-Sway Frames*, Joints in Structural Steelwork, edited by J.H. Howlet W.M. Jenkins and R. Stainsby, Pentech Press, London, United Kingdom, 1981

A survey on finite element modelling of steel end-plate connections

Naser NEMATI & D. LE HOUEDEC

LMS, Ecole Centrale de Nantes
1,rue de la Noe, 44072 Nantes, FRANCE

Summary

End-plate connections represent the joint types more widely studied in literature. This paper aims to present a complete review of finite element modelling of this joint type. After a general classification and presentation of the criterions for evaluating the models, according to the type of chosen finite element, the models will be discussed in four different categories : Plane stress, plate bending, shell and solid element models. Conclusion, perspective and a brief presentation of our laboratory 3D model, are included in the last paragraph.

1-Introduction

During the last ten years, experimental and theoretical researches have provided a lot of results and the concept of semi-rigidity has gradually entered in standards. The number of geometrical and mechanical parameters that can reasonably be expected to influence the joint behaviour is significant, but the experimental studies provide rather limited information. Thus, the Finite Element Method (F.E.M.), represents the most suitable tool for conducting a such exhaustive investigation. Furthermore, the results of this method could be a rational background for Eurocode standards. Because of important development of software and hardware in recent years, the progress in studying the semi-rigid connection has been fast and needs to be analysed.

Nethercot & Zandonini (1988) were the first to present a chronological analysis in this field . Among several joints, the end-plate connection deserves a special attention and we can find a lot of researches on it . Thus this paper is devoted to this type of joint and try to have a new point of view on the results published until 1995 .

There are several ways to classify the F.E. models : 1- Type of joint (bolted, welded), 2- Material behaviour law (elastic, elasto-plastic) 3- Finite element type (plane stress, plate bending, shell, solid), 4- Applied loading (monotonic, cyclic). For simplicity, we choose the third case for discussing the bolted joint and specially the end-plate connection . Referring to Fig. 1(standard end-plate joint), we notice that there are four possibilities for modelling a connection :
1- Modelling on the surface of Y-Z axis, using plane stress element. 2- Modelling on the surface of X-Z axis, using plane stress element. 3- Modelling on the surface of X-Y axis,

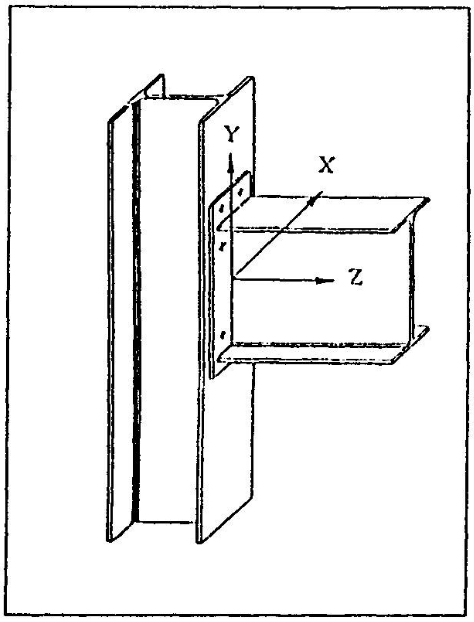


Figure 1- Standard end-Plate joint

Researchers	Year	Country	Model	Programme
Krishnamurthy	1976	USA	plane str.* & solid	
Ioannides & Tarpy	1979	USA	plate ben.**	
Ahuja	1982	USA	solid	
Ghassemieh	1983	USA	solid	
Jenkins & Tong	1986	UK	plate ben.	
Kukreti	1987	USA	plane str. & solid	
Colson	1989	France	plane str.	
Rothert	1992	Germany	solid	Prothec
Ziomek	1992	Poland	shell	Algor
Chasten	1992	USA	shell	Adina
Bursi	1993 & 94 & 95	Italy	shell & solid	Adina & Abaqus
Gebbeken	1994	Germany	plane str.	Prothec
Bahaari	1994	Canada	plane str.	Ansys
Sherbourn	1994	Canada	shell	Ansys
Masika	1995	Hungary	plane str.	
Nemati & Le Houedec	1996	France	solid	Samcef

* Plane stress elment ** plate bending element

Table 1- F.E. Models

using plate bending or shell elements 4- Three dimensional modelling . In this case we can use the shell element, the solid element or the combination of all kind of element (beam and shell elements, or shell and solid elements, etc...)

Table 1 gives the list of researches. From this table we can conclude that in recent years the European are more interested to this joint type. Furthermore, it appears more popular to use general codes.

2- Criterion of analysis

The comparison between the actual behaviour of joints and the behaviour of proposed models could be analysed from two points of view : 1- Final results 2- Phenomenon Modelling . It is obvious that the method for modelling the existing phenomena, changes the final result. The complete list of phenomena and the expected detail of results, allow us to evaluate the models .

2-1 Criterion based on final results

Moment rotation curve is the final product of a complex interaction between the member components. Thus a complete result includes the following characteristics :

A- Global behaviour of joint (i.e. moment-rotation curve) :

1- initial stiffness 2- plastic moment 3- hardening slope 4- ultimate moment
B- Local behaviour of components (force, displacement, stress, strain) :
1- bolts 2- plate 3-web and flange of beam 4-web and flange of column 5- welding etc...

2-2 Criterion based on phenomenon modelling

By taking into account the actual phenomena, simplification or neglecting some characteristics could be an important criterion . Generally, a complete model includes these characteristics :
A- Physical characteristics :
1- material and geometrical non-linearity and slope of hardening branch 2- hardening law (kinematic, isoparametric, mixed.) 3- effect of welding on material behaviour 4-buckling
B- Boundary conditions :
1- contact with or without friction : bolt head and plate, bolt shank and plate, plate and flange 2- type of loading : pure moment, shear force
C- Initial state :
1- initial displacement 2- preloading of bolts 3- residual stresses
D- Numerical characteristics :
1- type of element for each member 2- integration

3- Analysis

3-1 Plane stress element

In this study several types of plane stress element are used. Fig. 2 shows a summary of eight researches in this field with following explanations :

3-1-1 Modelling on the surface of Y-Z axis
This model consists to model the joint on the surface of beam and column webs. Hence, the biaxial applied moment is simplified by a linear distribution force on the surface of beam. All the geometrical depth must be taken into account in this surface. This object is achieved by two different approaches; in the first one, the thickness of elements is taken equal to the depth of related component in X direction. The second method uses an homogenisation technique by means of applying different Young's moduli for each member to find the proper thickness.

Krishnamurthy (1976),Fig. 2-a, *Kukreti (1987)*,Fig. 2-b,*Gebbeken (1994)*,Fig.2-g *and Bahari (1994)*,Fig. 2-c, used the first approach and *Colson (1989)*,Fig.2-d, applied the second . In almost of researches, the column is supposed rigid and the bolt head and welding effect are neglected . If we suppose the model of *Krishnamurthy* as a reference which used elastic-perfectly-plastic material, applied the bolt force using an initial displacement and modelled the contact by means of an iteration procedure for detecting the attachment, we can say that *Kukreti* followed exactly the same model. *Gebbeken* added the hardening material. *Bahari* added column, interface element for contact and bilinear and trilinear material models for plate and bolts, respectively .

The finding of this type of analysis is limited to : initial stiffness, ultimate moment, qualitative moment-rotation curve which can show the effect of thickness and the estimation of prying forces. However, with the parametric study of 2D model and finding the correlation factor between 2D and 3D, *Krishnamurthy* proposed his famous designe formula for AISC.

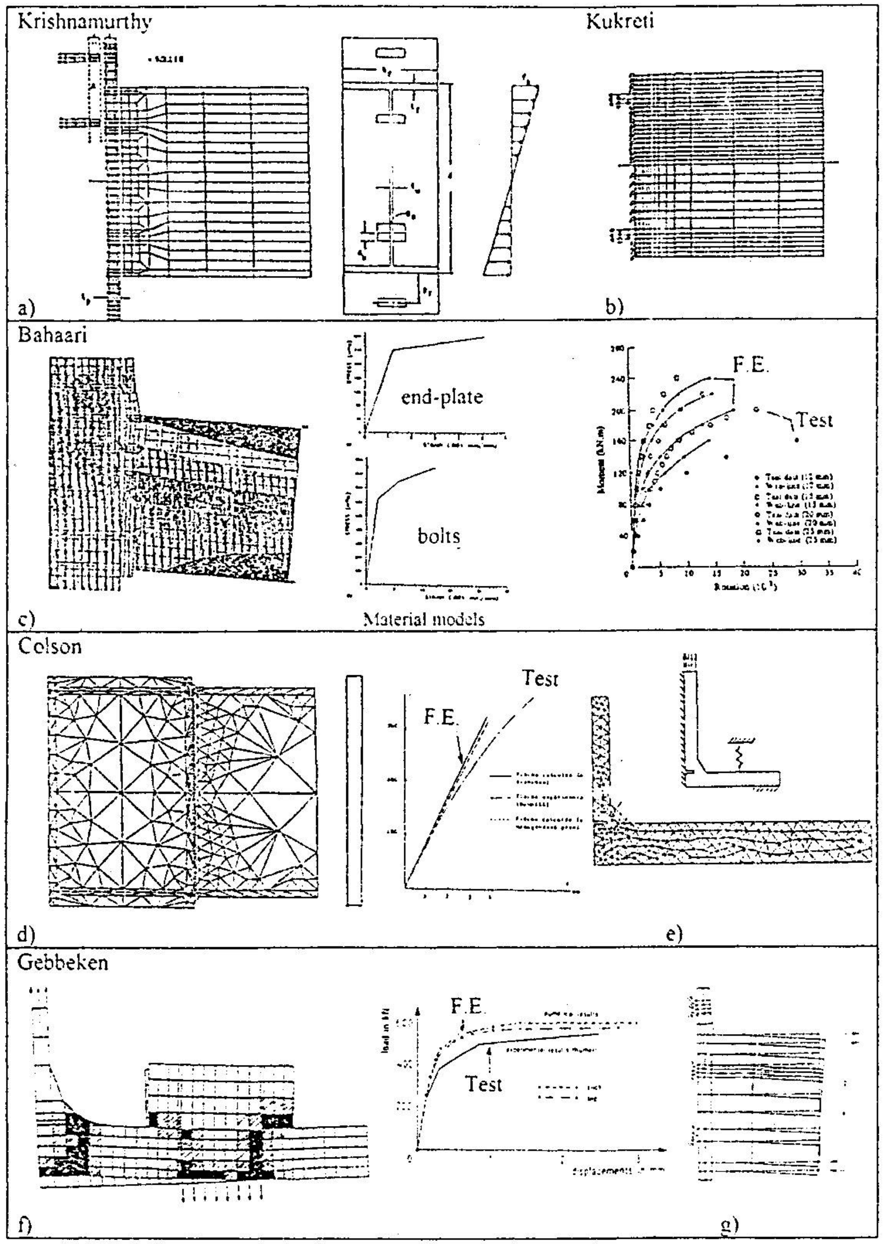


Figure 2-Plane stress models

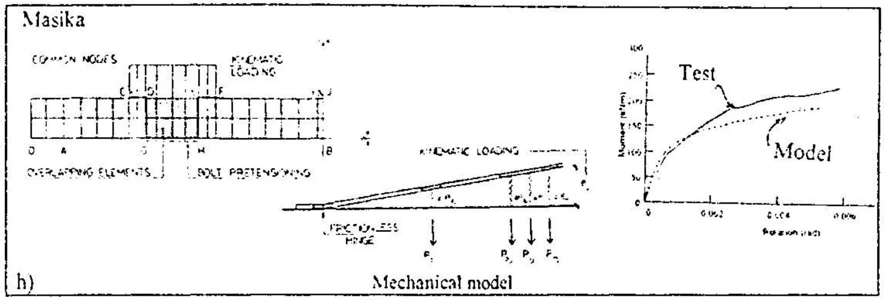


Figure 2-continue

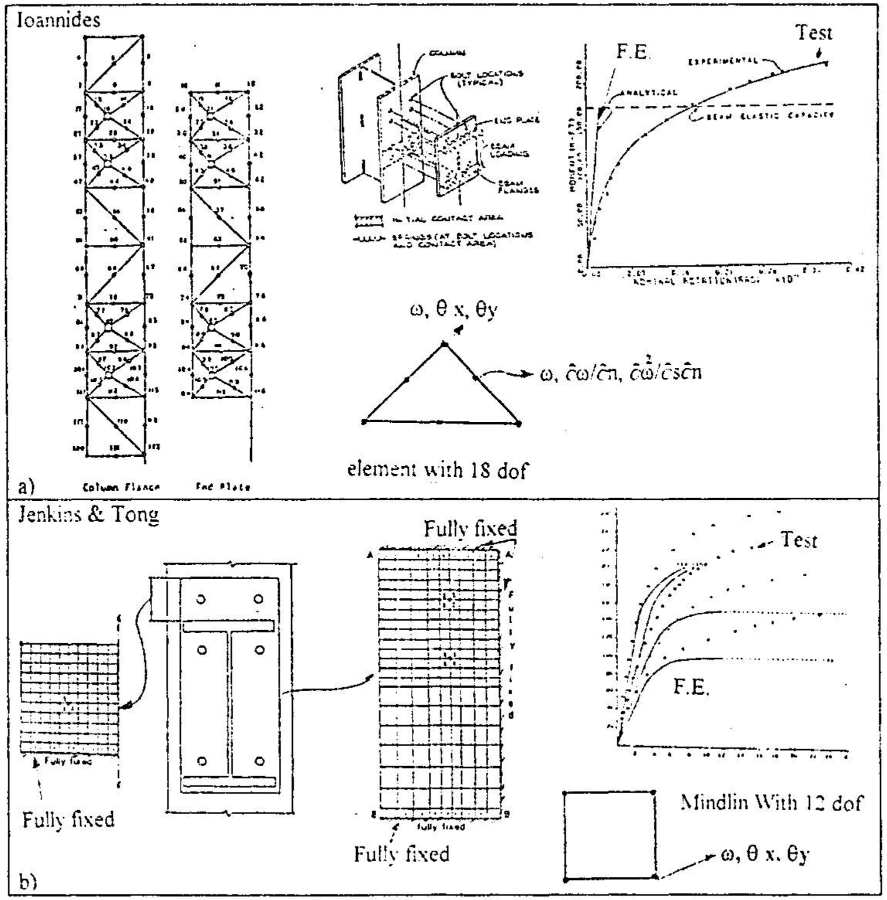


Figure 3-Plate bending models

3-1-2 Modelling on the surface of X-Z axis
This type of modelling consists to model the joint on horizontal plane. In this plane, end-plate and related beam web could be consider as a T-stub. The column flange and its web have the same shape (T-stub). Thus, by using the plane of symmetry, we can study a half of a T-stub .

Colson (1989) for finding the initial stiffness of an end-plate attached to a rigid foundation, used a new point of view . In this model, the bolts are replaced by a spring (Fig. 2-e). The assumption of non penetration welding is applied by introducing a crack between the welded line and flange . The results appear to be satisfactory .

Gebbeken (1994) studied the column flange with assumption of rigid end-plate. The procedure of study is the same as *Krishnamurthy*'s one by adding the bolt head and a frictionless slippage between the bolt head and flange . This study shows that introduction of contact between the bolt head and the related plate (here flange) changes seriously the results, Fig. 2-f .

Masika & al (1995) used a combination of F.E. method and mechanical model to find the global response of an end-plate haunched joint . The complete joint is divided to several independent T-stubs which are replaced by springs. The improvements of this method are: 1- combined kinematic and isotropic hardening laws 2- geometrical non-linearity 3- introduction of the failure mode by a given upper limit of the effective plastic strain. Unfortunately, there is no any direct comparison between T-stub results and test results to confirm this sophisticated modelling . Fig. 2-h shows a global behaviour of joint compared with experience .

In comparison with Y-Z surface (web) this model could be more realistic because the loading is really due to column web . The disadvantage is the neglect of interaction effect of neighbour T-stubs. Except some local results like initial stiffnes, effect of friction, slippage of bolt head, there is no direct result for joint. The spring model could represent an intermediate step to find the global response.

3-2 Plate bending element

In this method, the joint is modelled on the surface of the end-plate which involves neglecting the effect of beam .
Ioannides & Tarpy (1978) studied the interaction between end-plate and column flange . Each surface is modelled by a special triangular element (Fig. 3-a). The moment rotation is not satisfactory because the elastic-plastic material properties have not been included . However the effect of end-plate thickness on the column flange behaviour and displacement variation of end-plate are demonstrated qualitatively.

Jenkins, Tong & al (1986) studied the joint in two steps. In the first step, the modelling of Ioannides was followed (by adding the column web) and the prying force was estimated as much as 15% of applied force. In the second step, the end plate was divided on two independent plates with special boundary conditions illustrated in Fig.3-b .The moment-rotation curve is found by applying the displacement in the centre of bolts. By using the equilibrium equation and force-displacement of bolt (found in the last step) the applied moment could be found. The moment rotation results are softer than test results
We can notice that the nature of this type of modelling needs the simplification of boundary conditions, bolts, loading etc...and, consequently, it reduces the effectiveness of model.

3-3 Shell element

Because of existence of several components in a joint, it is not possible to use the shell element alone . For example the bolts are cylinders and in the perpendicular direction of end-plate. Thus, the use of other types of elements like beam element, solid element etc.. seems to be necessary. In this field we can find five independent researches which were done in last five years .

Devies & al (1990) designed a model for a frame with end-plate haunched connection . A complete frame is represented by shell element and the bolts with elastic beam element. In this research, the attention was paid to the global frame behaviour and to the possibility of local and member buckling. Hence the end-plate connection was not investigated in detail.

Ziomek & al (1992) conducted a study to find the effect of several numerical and physical parameters on one thin end-plate (thickness=12 mm, Fig.4-b) using the ALGOR programme. They concluded that modelling the material properties (choice of hardening law), bolts, finite element mesh as well as number of integration points, strongly influence results . The influence of geometrical non-linearity is rather small in this special case .

Chasten & al (1992) proposed a model with the code ADINA, for two end-plates with 19 and 25 mm of thickness in two phases. The first phase was the same as Ziomek's model by substitution the beam web with plane stress elements to find the transmitted force to end-plate. In the second phase, only the extended part of end-plate was studied. The bolts forces founded in this phase are reasonable(Fig. 4-a).

Bursi & al (1993)
This research is the same as the second phase of Chasten's model, but for extended and inner part of end-plate using the same code. The results of moment-rotation curve are more stronger than test result even for thinner end-plate with 12 mm of thickness.

The most complete research in this field belongs to *Sherbourn &t al (1994)*. By using the ANSYS code, a complete joint (with column) was modelled. The main difference of this work and the previous mentioned shell models is the modelling of bolts . To take into account the effect of bolt head and nut, they are idealised by isoparametric solid elements . The bolt shanks were substituted with six truss elements . The interface elements are used for solving the contact problem . The material properties are the same as their 2D works (*Bahaari & al*) . The results are satisfactory for thin end-plate . By increasing the thickness of end-plate, the shell element shows a little soft results (Fig.4-c). The prying action is plotted for some joints. It is important to mention that they used the nominal yield stress instead of actual one, which could influence the results i.e. more stronger moment-rotation curve.

3-4 Solid element

The solid elements are the natural selection for a 3D F.E. model, but the accuracy and the domain of application of model depend strictly on the simplifications, considered hypothesis and the type of element. In this field, there are seven works :

Krishnamurthy(1976) is again the first to use the solid element for end-plate connection but only with an elastic material . The maximum applied load is 0.6 time the yield stress . The element is a Levy's superparametric 33 dof, Fig.5-b. *Kukreti & al (1987)* used the same

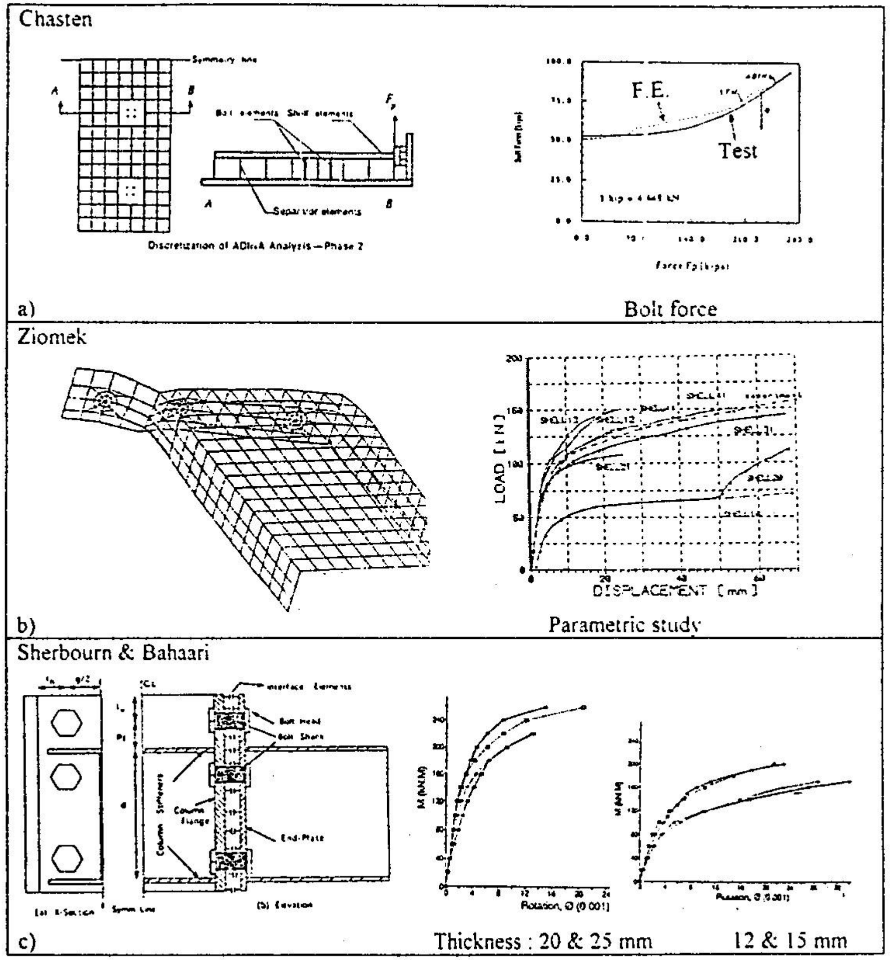


Figure 4- Shell models

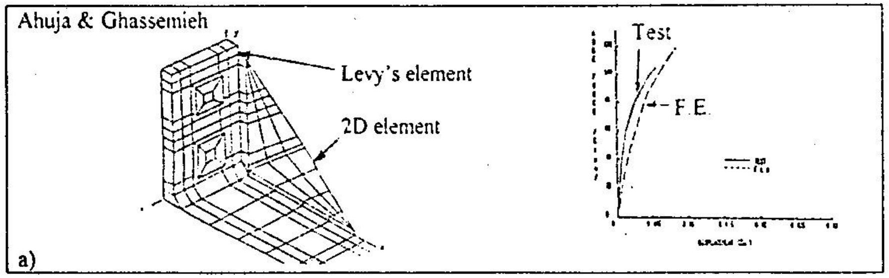


Figure 5- Solid models

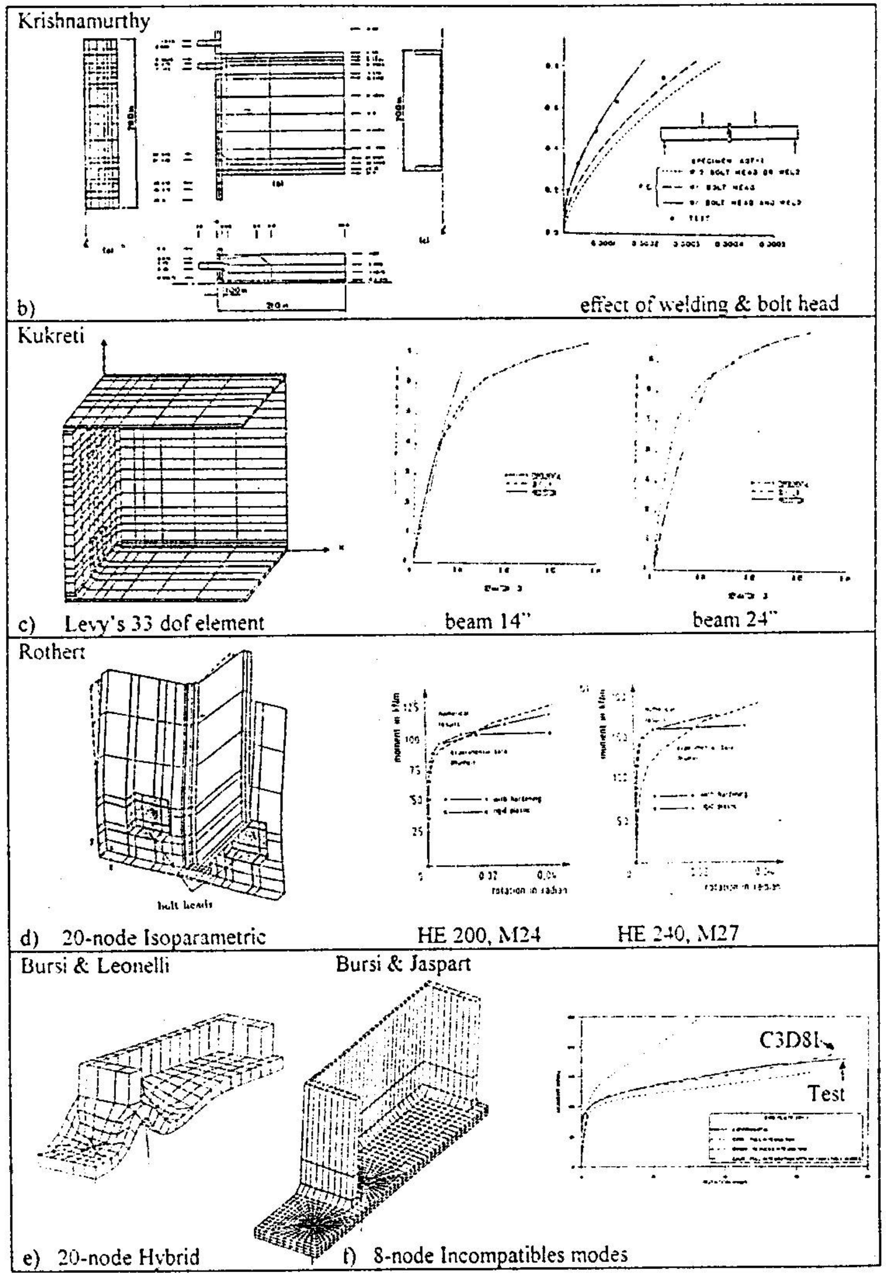


Figure 5- Solid models -Continue

element for flush end-plate joint by improving the material behaviour . In his model an elastic-perfectly plastic material was used. The comparison between experimental and numerical results shows that the errors of measured moment in the maximum applied displacement are between 6% and 30%. The maximum displacement is not too far from the plastic displacement of the joint (Fig.5-c) . In this type of modelling with simplification of bolts, welding, the contact approach, material properties etc... we can not judge about the element.

Rothert & al (1992) tried to model only the column flange . Because of two lines of symmetry, the one eighth of structure (one T-stub) was modelled(Fig.5-d). An isoparametric 20-node element is used . The thickness of end-plate was between 13.7 and 16.4 mm and one layer model was presented. The bolts and shanks are square shapes . The material nonlinearity is incorporated in model but the detail of work hardening rule is not clear. In reality, we can not model one eighth of flange and neglect the important interaction between several T-stubs. This is one of the important source of errors. Furthermore, there is a perfect adhesion between flange and bolt head. The effect and significance of non frictional contact between bolt head and end-plate are shown in 2D model.

Sedlacek & al (1994) during a short report about the research in Germany, proposed a 3D solid F.E. modelling for a single sided joint . Unfortunately the details are not available, The only curve shows the upper and lower boundaries results obtained from input parameter variation which compared with test result.

Bursi & al (1994) proposed an other model for a single side joint . In this model a 20-node hybrid solid element with one layer modelling of 12 and 25 mm thickness is used . The bolts are simplified and the standard two node beam elements to model shank and heads are used. The real beam is neglected and only a small portion of it is modelled . In this simulation, 27 Gauss integration points are used . The results are stronger than reality . For reducing this effect and this reality that non of the gauss points are located at the boundaries of the element (e.g. on the top and bottom surface of the end-plate), and both the yield and ultimate stress scaled to 0.77 of their actual values . With this correction, the result in hardening branch of moment-rotation curve is more softer than experience. In 1995, *Bursi & Jaspart* improved the last model by substituting the element with a 8-node incompatible modes(C3D8I in ABAQUS), and one layer element in thickness with three layers. Furthermore, a part of beam was added to the model. They presented the result for the 12 mm end-plate thickness which agreed with experience (Fig.5-e).

Nemati & Le Houedec (1996) proposed another model based on the SAMCEF code. The element is a 20-node isoparametric solid element with 8 points of Gauss. The most important characteristics of this model are; 1- a three layers mesh for end-plate is used 2- the bolts are modelled with solid element in circular shape, and there is contact between bolt head and end-plate 3- the effect of welding on heated zone which changes the yield point of material is taken into account 4- a bilinear hardening rule is used with actual yield stress. The slope of hardening branch of stress strain curve is equal to 2.5% of Young's modulus. The simulation is made for a joint with 18 mm of end-plate thickness. The results are satisfactory for monotonic loading(Fig.6). The same mesh is used for thin end-plate but the results appears to be softe. However the result with the same mesh but only one layer for thin end-plate and 4% slope for hardening branch of material, is acceptable.

As we mentioned before, the simplified models are not suitable to judge about the capacity of finite elements in inelastic behaviour. Furthermore, it is difficult to justify the use of only one layer element for thick end-plate in inelastic range, the neglect of the beam and modelling only one part of connection and extend the result to a complete joint.

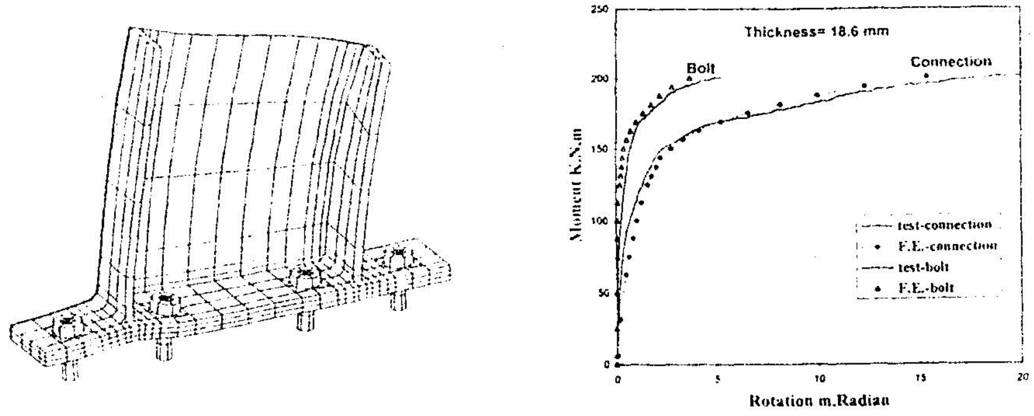


Figure 6- The model of Nemati & Le Houedec

4- Conclusions

With progress of softwares and hardwares in recent years, and today's industrial requirements, we can conclude that :

1- The model of plane stress and plate bending are not sufficient to have a complete understanding about end-plate joints.

2- The shell element model, reached at a good level but is limited to the very thin end-plate connections which are not really industrial. Furthermore the capability of shell element in cyclic loading and using the actual yield stress are in question.

3- The solid element model, seems to be the best solution for the problem, but for evaluating the capability of different solid elements, we need a complete model. The general programmes, allow us to model almost of all mentioned phenomena with minimum simplification. However the library of available elements in some of these codes seems not sufficient.

4- The following requirements are not incorporated in solid models until now :

a- actual loading (applied in the end of beam) b- residual stress c- lack of fit (we need new tests with correctly measured lack of fit) d- introduction of failure mode e- buckling effect which is very important specially in cyclic behaviour.

5- The following requirement are expected from solid F.E. models :

a- the contribution of each phenomena in final results b- parametric study c-cyclic behaviour of end-plate joint

6- the following problems are open to discussion :

a- is it necessary to use one type of element for all type of end-plate thickness ? The authors are convinced that the behaviour of very thin end-plates (however they don't seem industrial) are different comparing with thick ones. In thin connections we have an excessive deformation and all classical elements don't work in this situation.

b- In many models, the input data for material characteristics are artificial, for example:

I- the yield stress is not exactly the measured datum. The nominal or scaled values are used

II- The work hardening is perfect elastic-plastic or an arbitrary slope for a bilinear or trilinear curve is chosen.

References

1-Ahuja V. "Analysis of stiffened end-plate connections using the F.E. M." *Thesis submitted in partial fulfillment of the requirement for the degree of MSc, Uni. of Oklahama, USA, 1982*

2-Bahaari M.R. & Sherbourn A.N."Computer modelling of extended end-plate bolted connections,"*Computer and Struct. 1993*

3-Bursi O. & Leonelli L. "A finite element model for the rotational behaviour of end plate steel joint, " *Proc. of the second state of the art workshop, COST C1, Prague, October 1994*

4-Bursi O. & Leonelli L."Semi-analytical and F.E. based model for the rotational behaviour of end plate steel connection," *C.T.A. Collegio dei Tecnici dell'Acciaio, October 1993, Italy*

5- Bursi O. & Jaspart J.P. "A refined F.E. model for Tee-stub and end-plate steel connections," *C.T.A. Collegio dei Tecnici dell'Acciaio, October 1995, Italy*

6-Chasten C.P. & Lu L.W. & Driscoll G.C."Prying and shear in end-plate connection design, " J. *Structural Engineering, 118, No.5, 1992 pp. 1295-1311*

7-Colson A. "Modelisation des conditions aux limites de liaison et d'assemblages en mecanique des structures metalliques "*These de doctorat d'etat, Univ. Pierre et Marie Curie, (Paris VI), France, 1984*

8-Davies J.M. & Engel P. & Liu T.T.C. & Morris L.J."Realistic modelling of steel portal frame behaviour,"*The structural engineering, 68, No.1, 1990*

9-Gebbeken N. & Rothert H. & Binder B. "On the numerical analysis of endplate connection, " *j. Construct. Steel research, 30, 1994, pp. 177-196*

10-Ghassemeih M. "Inelastic F.E. analysis of stiffened end-plate connections "*Thesis submitted in partial fulfillment of the requirement for the degree of MSc, Uni. of Oklahama, USA, 1983*

11-Ioannides S. & Tarpy T.S."F.E. Analysis of unstiffened beam to column end plate connections,"*Proc. Third Int. Conf. in Australia on F.E. M. Univ. New South Wales, 1979*

12-Jenkins W.M.&Tong C.S.& prescott A.T."Moment-transmitting endplate connections in steel construction, and a proposed basis for flush endplate design,"*The Struct.Engr., 64A(5) 1986*

13-Krishnamurthy N.& Graddy D."Correlation between 2- and 3D F.E. analysis of steel bolted end-plate connections,"*Computers & structures, 6, 1976, pp. 381-389*

14-Kukreti A.R. & Murray T.M. & Abolmali A."End-plate connection moment-rotation relationship, "*J. Construct. Steel Research, 8, 1987, pp. 137-157*

15-Masika R.J. & Dunai L."Behaviour of bolted end-plate portal frame joints," *J. Construct. Steel Research, 32, 1995, pp. 207-225*

16-Nethercot D.A.&Zandonini R."Methods of prediction of joint behaviour:beam-to-column connections,"*Structural connections, stability and strength, Narayanan R. ,ed., Elsevier, 1989*

17-Nemati & Le Houedec "The analysis and 3D F.E. simulation of steel bolted end-plate connections,"CST96, Proc. of The Third Int. Conf. on Computational Technology, Budapest, August, 1996

18-Rothert H. & Gebbeken N. & Binder B. (1992) "Nonlinear 3D finite element contact analysis of bolted connections in steel frames,"*Int. J. Num. Meth. Engng. , 30, 1992, pp.303-318*

19-Sherbourne A.N. & Bahaari M.R."3D simulation of end-plate bolted connections,"*J. Structural Engineering, 120, No. 11, 1994, pp. 3122-3136*

20-Sedlaceck G.& Weynand K.&Kong B.S."Research in Germany in the field of semi-rigid connections," *Proceeding of the second state of the art workshop, Prague , October, 1994*

21-Ziomek D. & Tomana A. & Waszczyszyn Z."F.E. modelling of end plate connection versus experimental results," *Proceeding of the first state of the art workshop, COST C1, 1992, France*

Finite element-based models for the analysis of bolted beam-to-column steel connections

Oreste BURSI
Research Associate
Doctor Engineer
Trento,
ITALY

Oreste Bursi, born 1959, received his Mechanical Engineering Degree at the University of Padua, Italy. Since 1986, he has been at the University of Trento. His research interest is devoted to elastodynamics, structural plasticity and, finally, analysis and modelling of the behaviour of steel and steel-concrete composite structures.

Jean-Pierre JASPART
Research Associate FNRS
Doctor Engineer
Liège,
BELGIUM

Jean-Pierre Jaspart, born 1962, got his Civil Engineering Degree in 1985 and his Ph.D Degree in 1991 at the University of Liège. He is the author of more than 60 papers on steel connection and frame design, one of the members of the Drafting Group of EC3 Annex J and Chairman of the COST C1 Working Group on "Steel and Composite Connections".

Summary

This paper presents parts of the results of a study devoted to the analysis of bolted steel connections by means of finite elements. In particular, the paper introduces elementary tee-stub connections which are endowed with different plastic failure mechanisms and can be adopted as benchmarks in the validation process of finite element software packages. The comparison between computed and measured values permits the effectiveness and the degree of accuracy of the proposed finite element models to be highlighted.

1. INTRODUCTION

Bolted connections are widely used in steel frames as simple or moment-resistant connections between steel members. Usually, they are designed to achieve "pinned" or "rigid" connections, though steel structures need often to have the lowest level of detailing compatible with design requirements. A solution to this problem has been obtained with the recent semi-rigid design philosophy. This approach provides greater freedom than simple or fully-continuous design because the properties of the connections are treated as variables in design, to be chosen to meet the individual requirements of each project. Hence, the knowledge of the joint response and how it affects frame performance becomes a prerequisite to the practical use of semi-rigid design [1].

The potential economic implication of connections on frame design and fabrication is also realized by modern codes, such as LRFD [2] and Eurocode 3 [3]. In particular, Eurocode 3 includes application rules in order to define explicitly the joint behaviour [4]. In this context, the finite element technique can represent a rational supplement to design.

The study presented in this paper has a twofold purpose: *(i)* to introduce elementary tee-stub connections which can be used as benchmarks in the validation process of finite element software packages for bolted connections; *(ii)* to present a rational approach to calibrate a finite element model able to reproduce the elastic-plastic behaviour of elementary tee-stub connections.

The paper describes the behaviour of elementary tee-stub connections and their simulations with the LAGAMINE software package [5] by means of bricks and contact elements. As explained in section 2, this work has followed by the simulation of the elastic-plastic behaviour of more realistic end plate connections to the ultimate limit state by means of the ABAQUS code [7,8], including both end plate-foundation and end plate-bolt interaction phenomena. Because of the limited number of pages, this part of the study (see [6]) is not described in the present paper.

2. Approach of the Study

Nowadays, latest generation research and commercial finite element codes are capable to simulate almost all the complex phenomena affecting the connection response (three-dimensional behaviour, combined non linear phenomena like material and geometrical non-linearities, friction, slippage, contact, bolt-plate interaction and fracture, ...). However, still difficulties remain to the numerical analyst which has to choose appropriate finite element models able to provide an accurate representation of the physics with the lowest computational cost. Choice of mesh, node number, integration point number through the element thickness and time-step size for constitutive law integration depend upon resources, problem, geometry, type of loading and required accuracy .

To shed light on these problems, elementary non preloaded and preloaded tee-stub connections have been tested in laboratory by Jaspart [10] and Bursi [11]; they are proposed as benchmarks in the validation process of finite element software packages. These benchmarks have been simulated in a large displacement, large rotation and large deformation regime with the LAGAMINE software package [5], by means of bricks [12] as well as contact elements [13]. The choice of these elements as well as related aspects have been commented upon and the numerical results have finally been compared to the experimental ones, thus assessing the reliability of the finite element models.

A calibration phase has then been described in which specific elements of the ABAQUS library [8] have been chosen on the basis of test data as well as LAGAMINE [5] simulations. Then, additional simulations have been performed to validate an assemblage of beam elements, labelled spin, which is intended to reproduce in a simple, yet accurate manner, the bolt behaviour. Finally, the ABAQUS [7] code has been used to simulate the elastic-plastic behaviour of realistic end plate connections to the ultimate limit state. The comparison between computed and reference values in each phase has allowed to highlight the effectiveness and degree of accuracy of the proposed finite element models.

A detailed information concerning this work may be found in [6] and [9]. The main aspects of the first part (study of the benchmarks by LAGAMINE) are presented here below.

3. Simulation of tee-stub connections

3.1 Experimental data used as references

In order to acquire basic experimental data, elementary tee-stub connections proposed by Jaspart [17] and, afterwards, by Bursi[18] within the Numerical Simulation Working Group of the European research project COST C1 "Civil Engineering Structural Connections" were tested to collapse. These specimens reflect different geometrical and strength parameters as well as bolt prestressing conditions. In the sequel, the specimens are labelled T1 and T2 and are represented with their geometrical characteristics in Fig. 1a and 1b, respectively. The stub beam specimens were obtained from the same IPE300 and HE220B profiles, respectively, to allow a direct comparison of performances among the specimens. Furthermore, they were designed purposely to fail according to the so-called Mode 1 and Mode 2 collapse mechanisms described in Eurocode 3 [3]. Fasteners were M12 grade 8.8 bolts. Within T1 and T2 specimens both non-preloaded and prealoaded bolts were used. As a result, the overall test program comprised four tests.

A full detailed description of the test data and results is given in [9].

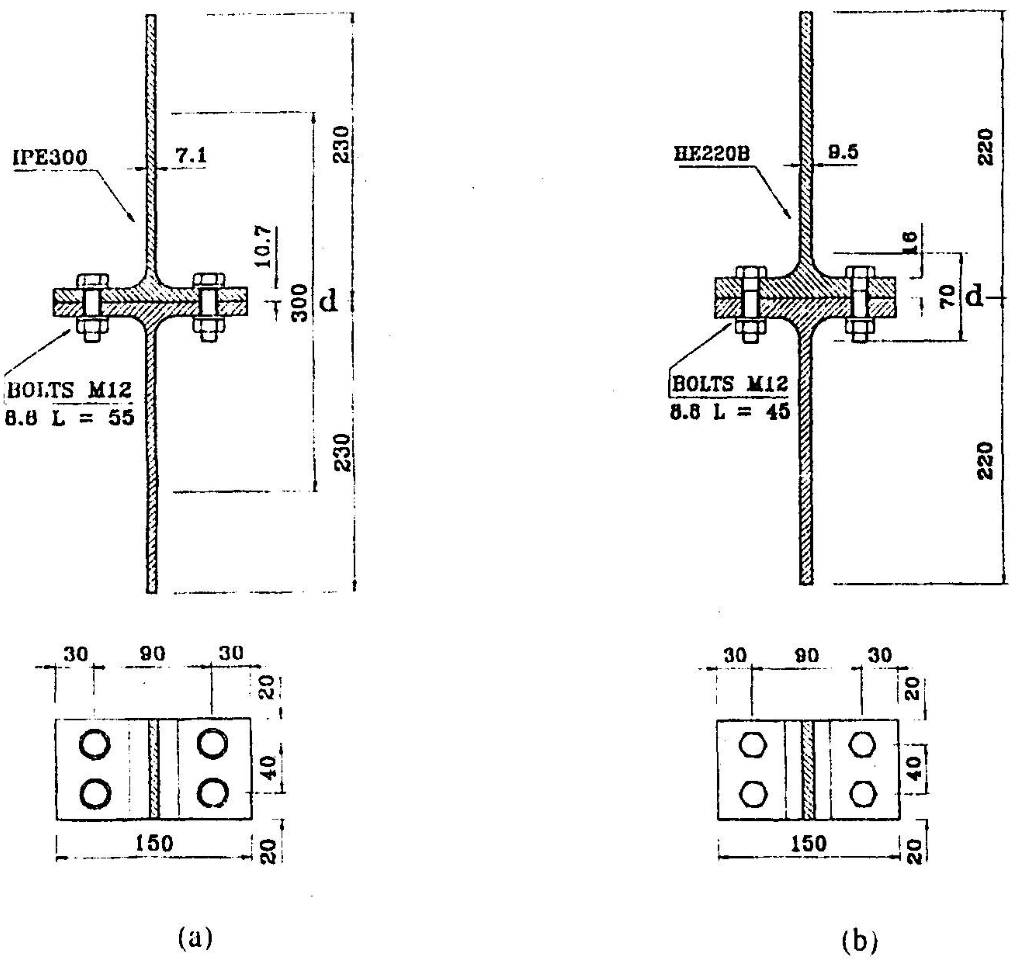


Fig. 1. Test specimens: (a) tee-stub connection T1; (b) tee-stub connection T2

3.2 Types of finite element

The LAGAMINE finite element package, developed at the MSM Department of the University of Liege to simulate metal-forming [5], i.e. processes characterized by large displacements, large rotations and large strains, has been used to predict the behaviour of the isolated tee-stub connections described above.

These elementary connections are proposed as benchmarks for finite element modelling, because they embody many typical features of bolted connections. In particular, the material discontinuity within these assemblages determines a relative movement of their constitutive components. In addition, these components are subjected to yielding while bearing of fasteners and elements determines stress concentrations or prying forces.

In order to minimize the number of modelling assumptions, those complex 3D phenomena have been reproduced by adopting both hexahedra [12] and contact [13] finite elements implemented in LAGAMINE. These are briefly described hereafter but the interested reader will find more information in [9].

Hexahedra elements are more popularly known as bricks. In the simulations of the tee-stub connections performed with LAGAMINE, use has been made of the three dimensional mixed brick element called JET3D [12] in which an assumed strain field within a mixed-multifield variational principle able to eliminate spurious energy-modes and a geometry dependent parameter set to control shear locking have been embodied.

The contact and distribution of interface stresses between two bodies are unknown during a contact process, and therefore, the contact problem turns out to be highly non-linear and with unknown boundary conditions. This phenomenon is simulated in LAGAMINE with contact elements which describe topologically surfaces and are located on the boundaries of solid elements. The contact condition is guaranteed by a penalty technique [13] which requires an optimal value for a penalty parameter. This parameter can be interpreted as the stiffness of a virtual spring between two bodies. As a result, contact constraints are satisfied only in the limit for an infinite penalty value. However, a too large penalty value can engender ill-conditioning of the stiffness matrix. Thus, its optimum value is traced when there is only a slight change in the results for an additional increase of the penalty parameter or, when the penetration reaches limited values.

3.3 Plate and bolt discretization

In Li's work [14], it is shown that, for bending-dominated problems, at least three-layers of JET3D brick elements have to be used to capture the stiffness and strength behaviour of a structure with a good accuracy.

In the examined specimens as well as in bolted connections, in general, bolts behave in a 3D fashion. Hence, they have also been modelled in the benchmarks by means of JET3D brick elements. Nevertheless in order to reduce the number of contact planes, washers have been considered attached to bolt heads and bolts have been assumed to be symmetric. To comply with these assumptions, the additional flexibilities provided by the nut and the threaded part of the shank have been incorporated into an effective bolt length according to the Agerskov's model [15]. That model is based on the following equation:

$$\frac{E A_b}{K_1 + 2K_4} = \frac{B}{\Delta l_b} = \frac{E A_s}{L_{eff}} \tag{1}$$

where A_b and A_s indicate the gross cross-section and the tensile stress area, respectively, B and Δl_b define the bolt force and the corresponding bolt elongation while the effective length L_{eff} is unknown. K_1 and K_4 are parameters which can be obtained readily from the bolt geometry shown in Fig. 2. In detail, the following relations hold:

$$K_1 = l_s + 1.43 l_t + 0.71 l_n \qquad K_4 = 0.1 l_n + 0.2 l_w \tag{2}$$

By noting that the threaded part of the bolt shank triggers off tensile yielding and failure, the bolt shank is reproduced with a cylinder of cross-section area A_s. With this assumption, L_{eff} can be obtained readily from Eq. (1).

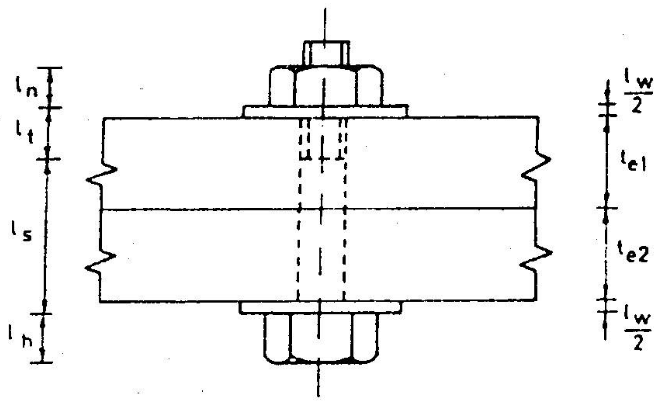


Fig. 2 Bolt geometry

3.4 Finite element results

Finite element analyses covered all four specimens. However, because of the limited number of pages, the results are described accurately for the preloaded specimen T1 only in the remainder of this section.

The displacement field at the plastic failure state traced by the finite element analysis is shown in Fig.3.a for the preloaded specimen T1. One can observe how the model is able to reproduce the flange kinematics and the relative movement between flange and bolt head. The corresponding distribution of von Mises equivalent stresses is reported in Fig. 3.b. The large stress fields in the flange near the bolt hole and close to the radius of fillet identify two yield lines which govern the kinematic mechanism observed at yielding. This failure mechanism agrees with the one predicted by Eurocode 3 (Mode 1).

The accuracy of the finite element model can be quantified by superimposing the computed load-displacement F - Δd relationships upon the measured one, as shown in Fig. 4. From the comparison one can observe the good accuracy of the simulation. Only some discrepancy is evident at the onset of yielding, due to residual stress effects which determine a more gradual plastification of the specimen and which are disregarded in the model.

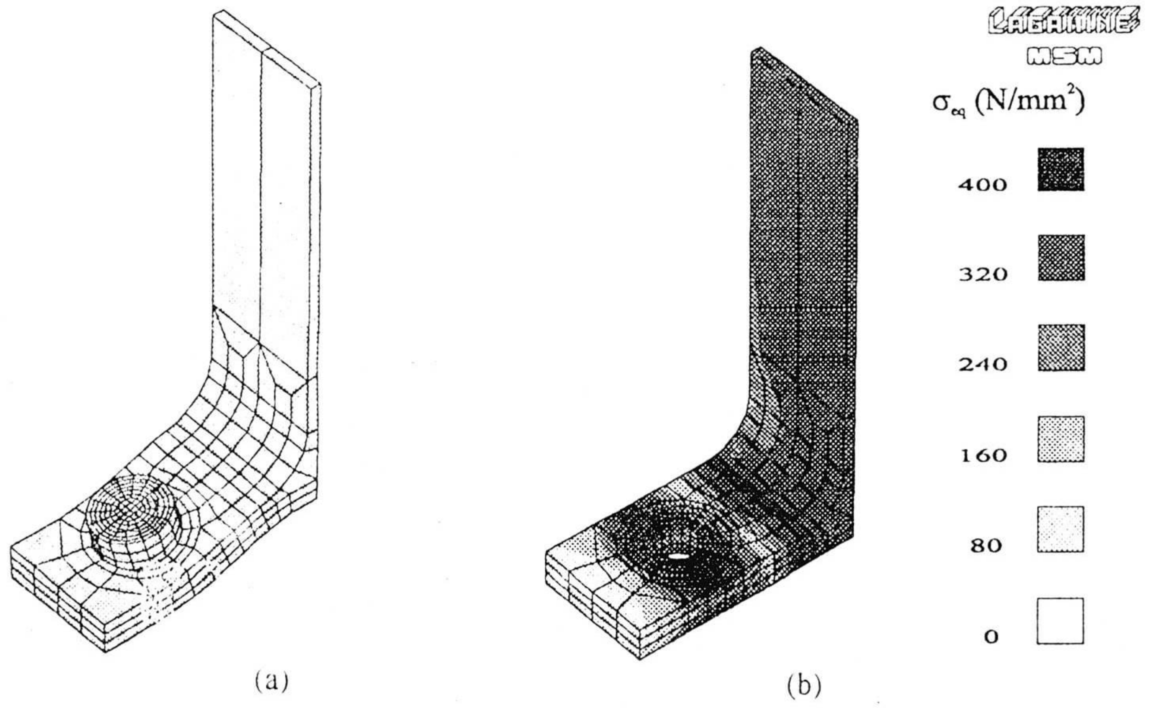


Fig. 3 Preloaded tee-stub T1 at the plastic failure state: (a) displacement field; (b) von Mises equivalent stress field

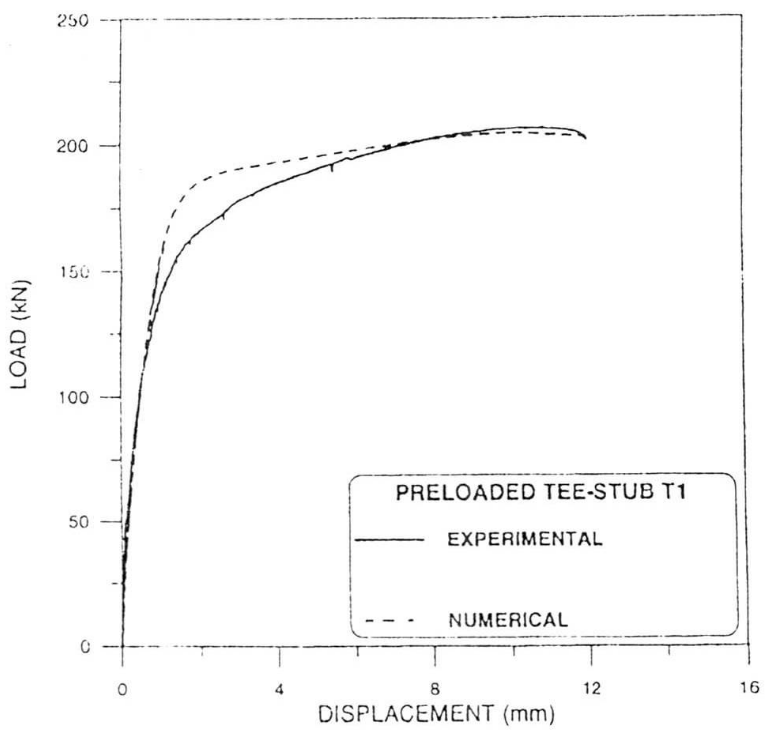


Fig.4 Experimental and predicted relative displacement Δd versus load F for the preloaded tee-stub T1

Once the finite element model is proved to be reliable, it can be used to generate information which cannot be provided from actual tests, because bolted connections appear to be highly redundant and confined physical systems. As an example, significant data can be obtained by plotting contact pressures developed between tee-stub flanges. Fig. 5 highlights the normal pressure distribution at the plastic failure state in the external part of the tee-stub. This distribution can be used to quantify the location and amplitude of prying forces.

The model can also provide detailed information on bolt behaviour. The evolution of von Mises stresses in each bolt and washer can be observed both at the preloaded state and at the plastic failure state of the tee-stub in Fig. 6.a and 6.b, respectively. Yielding can be observed in the bolt shank, indicating that also bolts participated in the plastic failure mechanism. In addition, from Fig. 6.b one can observe the stress level which affects the washers.

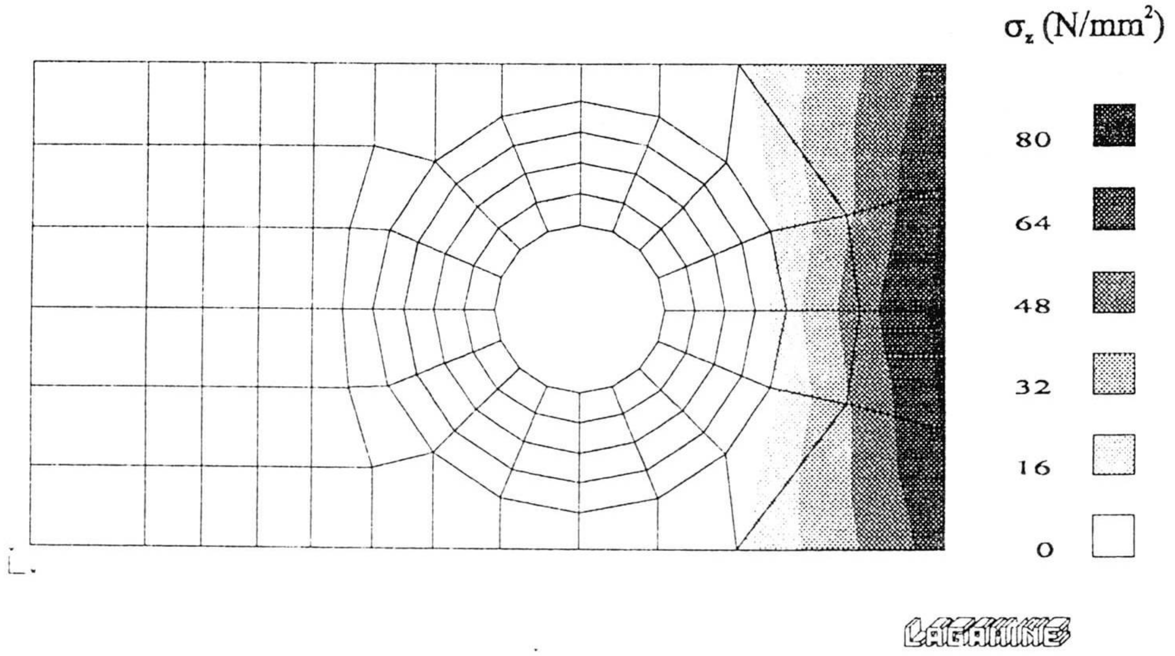


Fig. 5 Normal pressure distribution at the failure state for the preloaded tee-stub T1

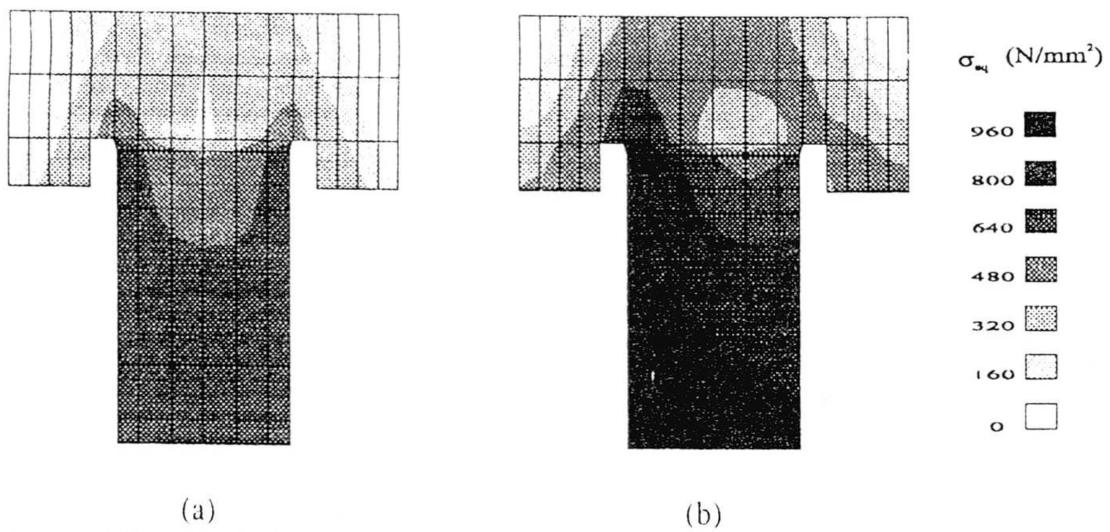


Fig.6 von Mises equivalent stress field of bolt in the preloaded tee-stub T1: (a) preloaded state: (b) plastic failure state

The curve giving the bolt axial force versus the applied total force is plotted in Fig. 7. From this relationship one can trace the evolution of the bolt force characterized by a first phase dominated by preloading effects followed by a growth of the bolt force due to prying effects. By means of this plot prying force values can be evaluated quite easily. In addition, bolt bending moment values can be evaluated too.

For brevity, the accuracy of the finite element model for the corresponding non-preloaded specimen is assessed by comparing the load-displacement relationship only. Such a comparison is shown in Fig. 8, where one can observe the accuracy of the simulation. Also for this case, a major discrepancy can be observed at the onset of yielding, where the actual plastification appears to be more gradual due to the residual stress effects.

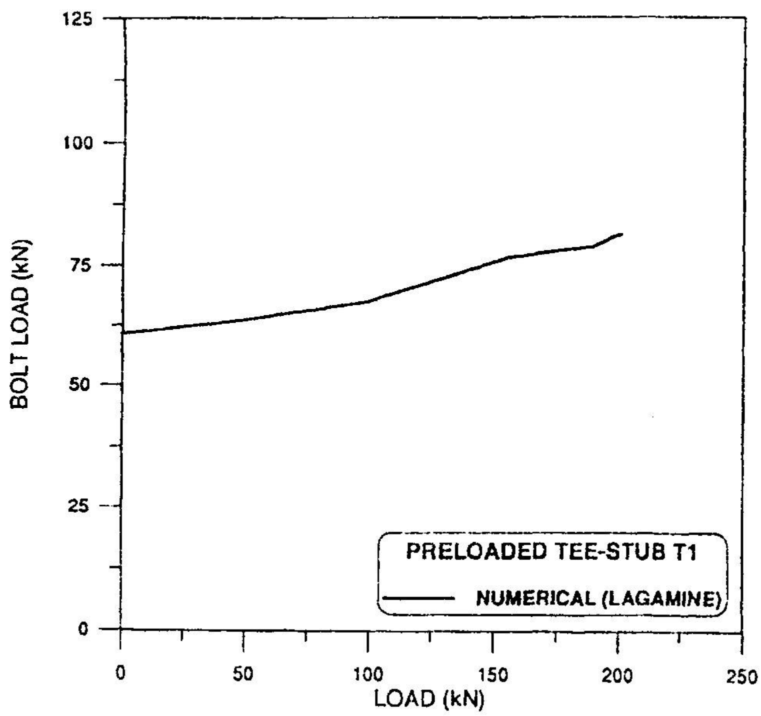


Fig. 7 Bolt axial force versus total force F for the preloaded tee-stub T1

4. Conclusions

Parts of results of a study devoted to the analysis of bolted steel connections by means of finite elements have been presented in this paper. Initially, two elementary tee-stub connections have been proposed as benchmarks in the validation process of finite element software packages for bolted connections. Then, a rational approach that leads to an accurate simulation of these connections by means of a three-dimensional finite element model has been suggested. The model which has been set with the LAGAMINE software package has been able to reproduce many of the characteristic phenomena embodied in bolted connections. The comparison between computed and measured values in each phase has highlighted the effectiveness and degree of accuracy of the proposed finite element models. The data obtained in this study are used in [6], where a simplified three-dimensional finite element model is set to reproduce the elastic-plastic behaviour of full bolted end plate connections by means of the ABAQUS code.

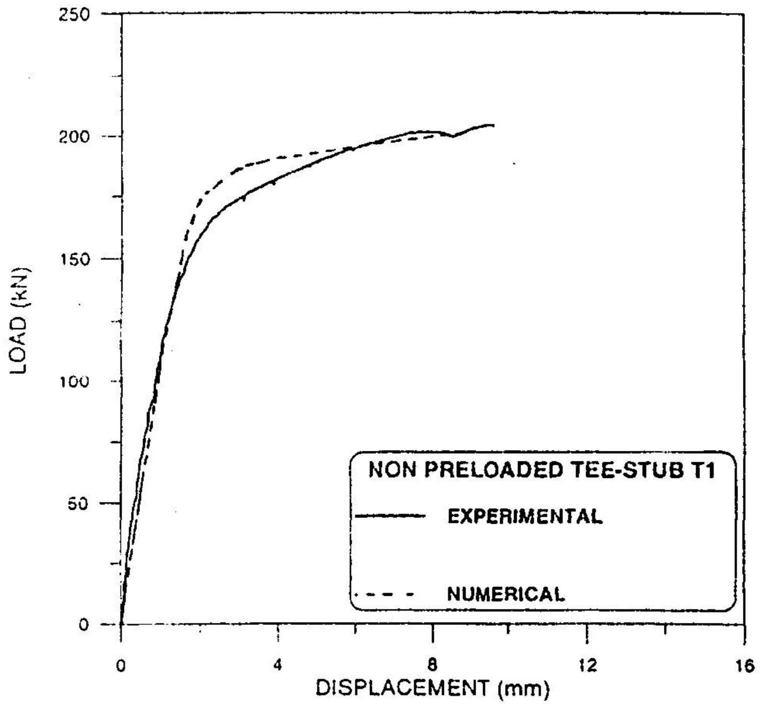


Fig. 8 Experimental and predicted relative displacement Δd versus load F for the non preloaded tee-stub T1

Acknowledgements

The work reported in this paper is granted by MURST of Italy. Specific support to the University of Trento and Liège is also provided by the SPRINT Contract RA 351 financed project "Steel and Composite Building Frames using Semi-Rigid Connections: From Research to Design Practice". The findings, observations and conclusions in this paper are, however, those of the writers. The skillful work of Dominique Hanikenne, Luca Leonelli, Françoise Taquet and the technical staff of Testing Laboratories is greatly appreciated.

References

[1] Technical Work Group 8.2,"Analysis and Design of Steel Frames with Semi-Rigid Joints", *European Convention for Constructional Steelwork*, publication n° 67, ECCS, Brussels, (1992).

[2] AISC, "Manual of Steel Construction - Load and Resistance Factor Design (1st edn)", *American Institute of Steel Construction*, Chicago, IL, (1986).

[3] Eurocode 3 - ENV 1993-1-1: "Design of Steel Structures Part 1.1: General Rules and Rules for Buildings", *CEN*, Brussels, February, (1992).

[4] Eurocode 3 - New Revised Annex J: Joints in Building Frames. *ENV 1993-1-1/pr A2*, (1994).

[5] Cescotto S., Habraken A-M., Radu J-P and Charlier R., "Some Recent Developments in Computer Simulations of Metal Forming Processes, *Proc. 9th Int. Conf. on Computer Methods in Mechanics*, 4, Krakow-Rytro, Poland, (1989), 19-52.

[6] Bursi O.S. and Jaspart J.P., "Calibration of a Finite Element Model for Bolted End Plate Steel Connections", *Journal of Constructional Steel Research*, (submitted).

[7] ABAQUS - Theory Manual. Version 5.4, Hibbitt, Karlsson & Sorensen Inc. (1994).

[8] ABAQUS - User's Manual. Vol. I & Vol II. Version 5.4, Hibbitt, Karlsson & Sorensen Inc. (1994).

[9] Bursi O.S. and Jaspart J.P., "Benmarchs for Finite Element Modelling of Bolted Connections", *Journal of Constructional Steel Research*, (submitted).

[10] Jaspart J.P., "Numerical Simulation of a T-stub - Experimental Data", *COST C1 Numerical Simulation Working Group*, Doc. C1WD6/94-09, (1994).

[11] Bursi O.S., "A Refined Finite Element Model for Tee-stub Steel Connections", *COST C1 Numerical Simulation Working Group*, Doc. C1WD6/95-07, (1995).

[12] Li, K.P., Cescotto, S., & Jetteur, P., "An Element with Hourglass Control for Large Deformation Analysis in Three Dimension", *Third Int. Conf. on Computational Plasticity: Fundamental and Applications*, Barcelona, (1992), 2021-2032.

[13] Charlier R. and Habraken, A.M., "Numerical Modellisation of Contact Phenomena by Finite Element Method", *Computers and Geotechnics*, 9, (1990), 59-72.

[14] Li K.P., "Contribution to the Numerical Simulations of Three-Dimensional Sheet Forming by Finite Element Method", Ph.D. Thesis, Chapter 2, Dpt MSM, University of Liege, Belgium, (1996).

[15] Agerskov, H., "High-Strength Bolted Connections Subject to Prying", *Journal of Structural Engineering Division, ASCE*, Vol. 102-1, (1976), 161-175.

Three Dimensional Nonlinear Finite Element Simulation of R/C Joints

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

Stefan KINAREV
Assistant 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, 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.

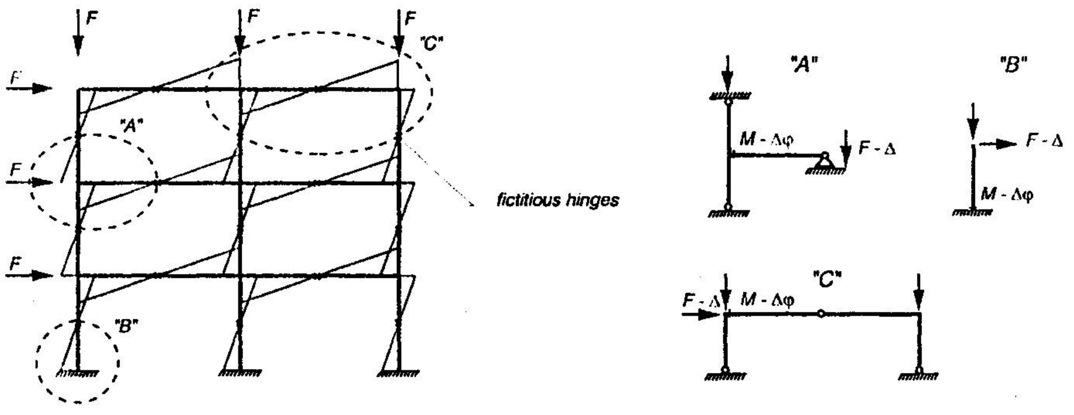


Figure 1. The bending moment diagram of a plane frame, loaded laterally and the identified substractures

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].

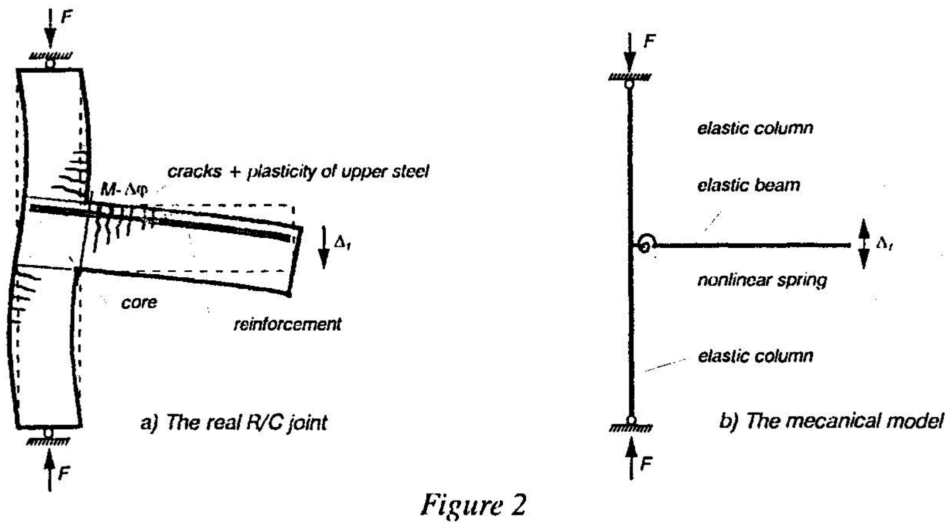


Figure 2

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.

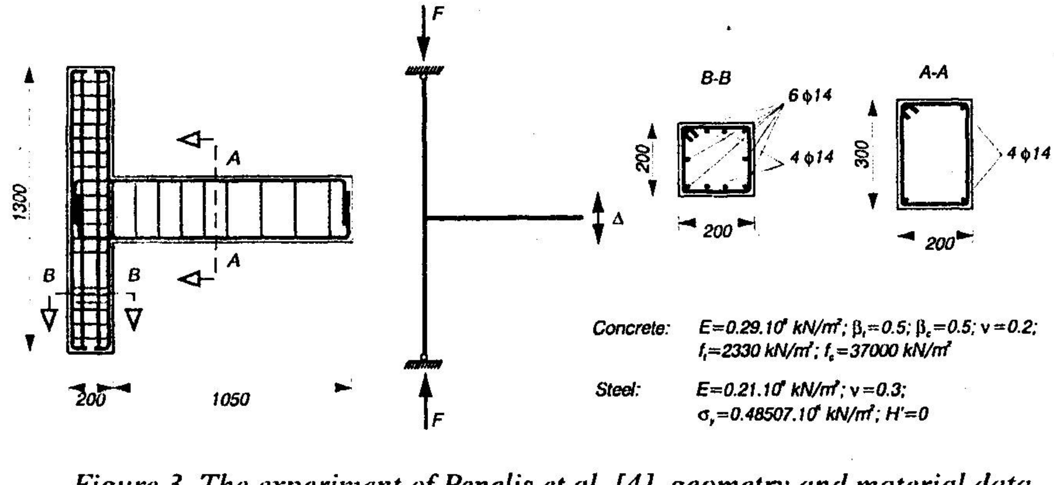


Figure 3. The experiment of Penelis et al. [4], geometry and material data

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.

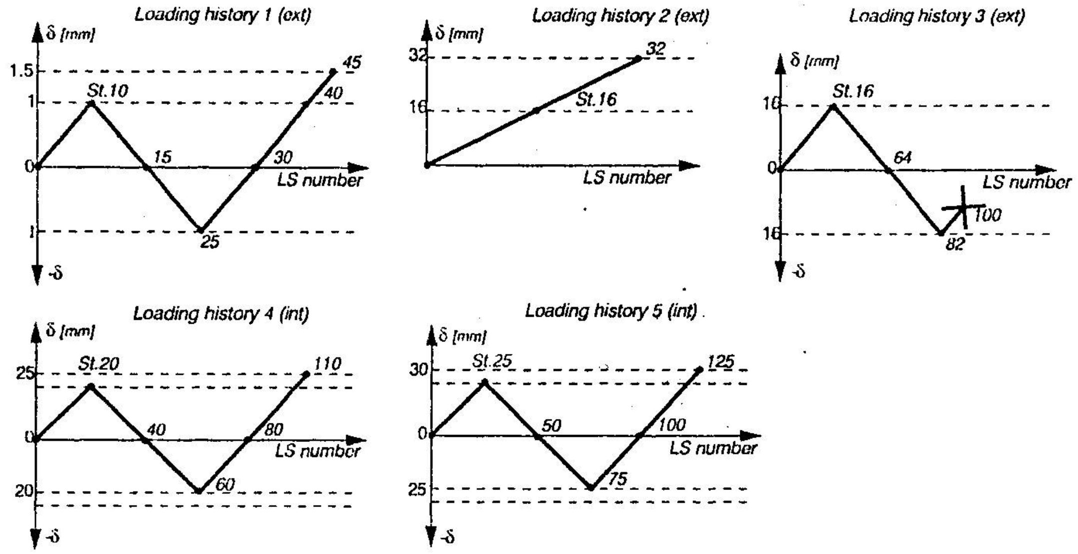


Figure 4. The loading sequence for the loading cases considered

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.

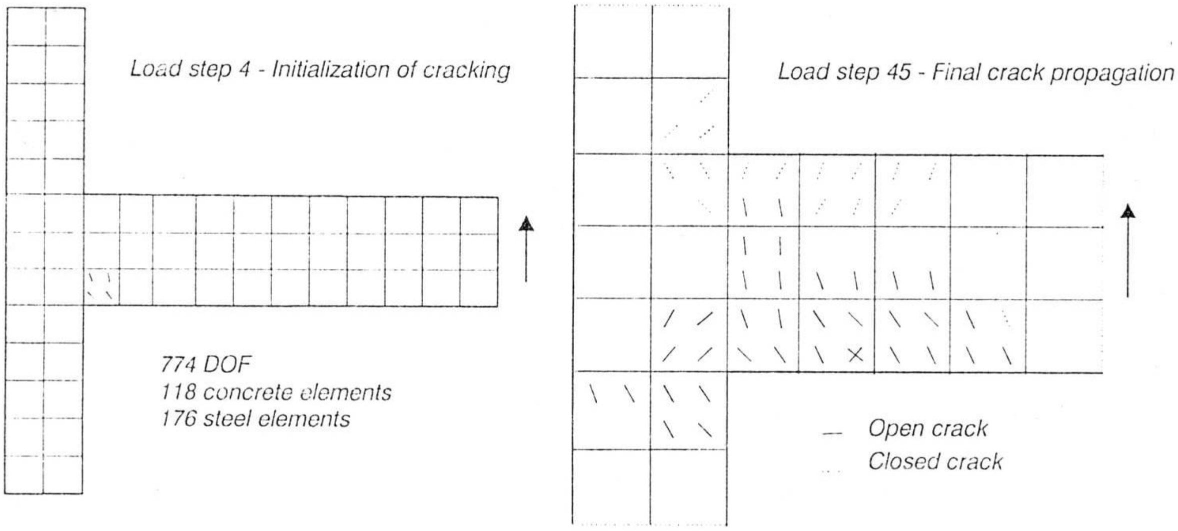


Figure 5. Loading case 1 - FE mesh data and tensile crack development at load step 4 (0.4 mm) and load step 45 (1.5mm)

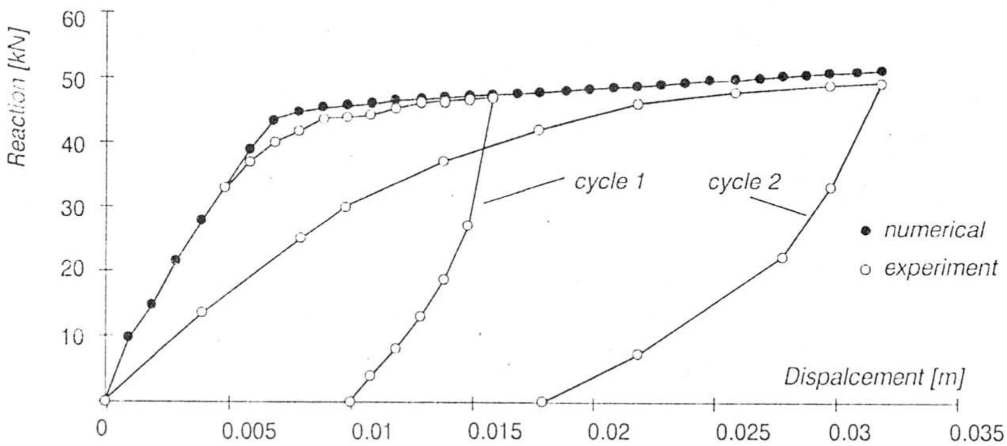


Figure 6. Loading case 2 - Reaction - displacement curve

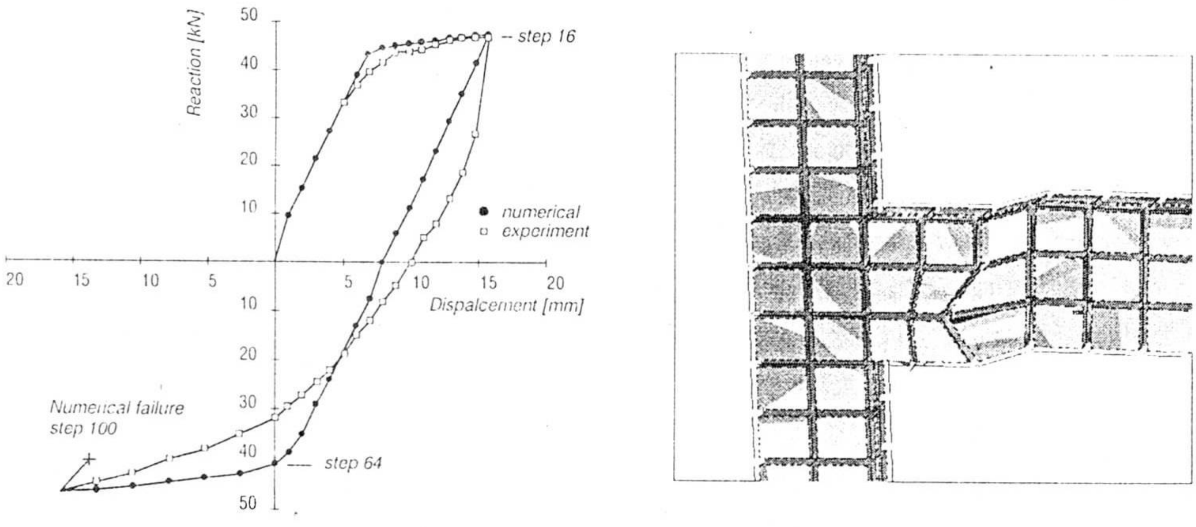


Figure 7. Loading case 3 - Reaction - displacement curve and collapse mechanism

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.

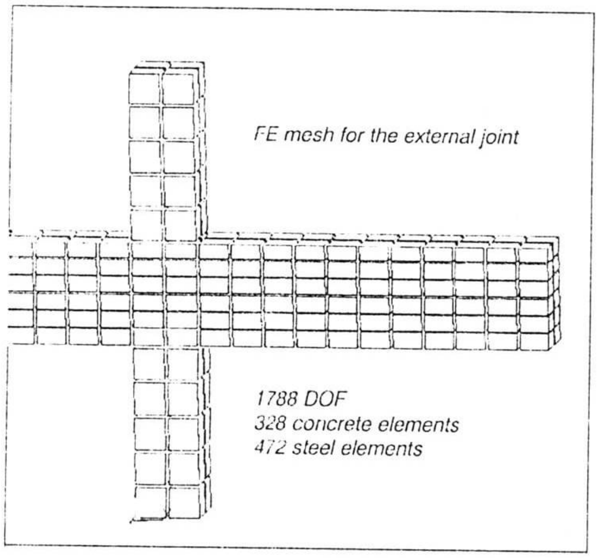


Figure 8. Internal joint - FEM data

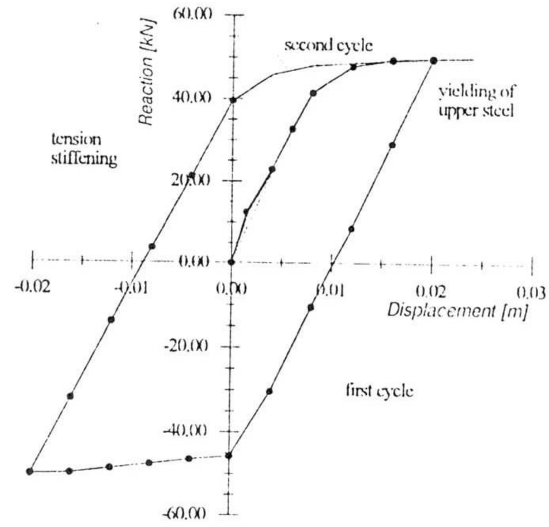


Figure 9. Loading case 4 - Reaction-displ. curve

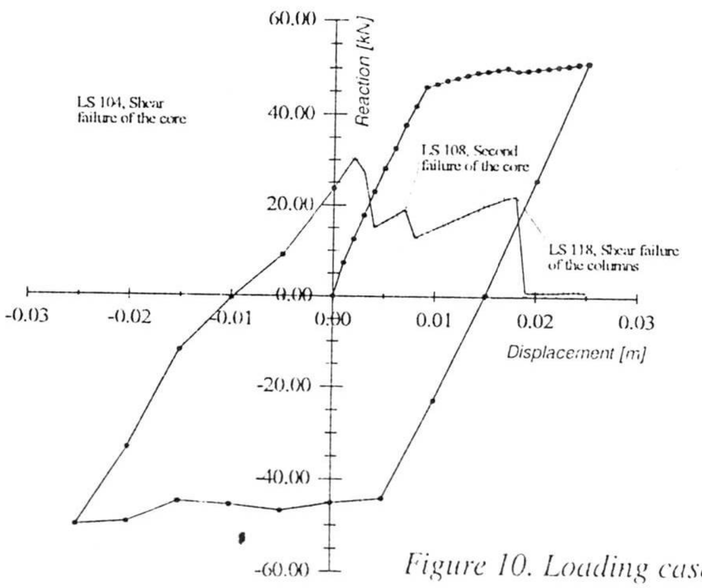


Figure 10. Loading case 5 - Reaction-displ. curve

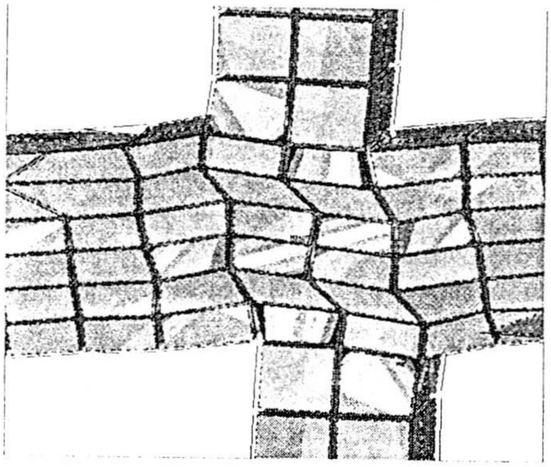


Figure 11. Shear failure of the core

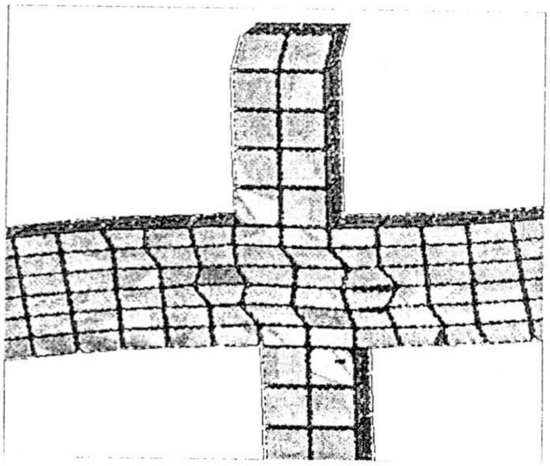


Figure 12. Final shear failure of the columns

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", Procc. 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", Procc. 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", Procc. 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.*