C.T.A. Collegio dei Tecnici dell'Acciaio
GIORNATE ITALIANE DELLA COSTRUZIONE IN ACCIAIO

RIVA DEL GARDA: 15 - 16 - 17 - 18 ottobre 1995

A REFINED FINITE ELEMENT MODEL FOR TEE-STUB AND END-PLATE STEEL CONNECTIONS

UN MODELLO ACCURATO AGLI ELEMENTI FINITI PER COLLEGAMENTI A T E COLLEGAMENTI FLANGIATI

O. Bursi[(1)] and J.P. Jaspart[(2)]

[(1)]Dipartimento di Meccanica Strutturale e Progettazione Automatica,
Università di Trento, Via Mesiano 77, 38050 - TRENTO

[(2)] Department MSM, University of Liege, 6 Quai Banning, B4000 - LIEGE

ABSTRACT

Within the activities of the European research project COST C1 "Civil Engineering Structural Connections", a three-phase strategy that leads to an accurate simulation of the moment-rotation behaviour of extended end plate steel connections is developed. In the first phase, elementary tee-stub connections characterized by different plastic failure mechanisms are simulated with a three-dimensional finite element model set by using the LAGAMINE software package. The second phase concerns the calibration of the general purpose code ABAQUS based on LAGAMINE, assumed as a reference code. In the third and on-going phase, the ABAQUS code is adopted to reproduce the elastic-plastic behaviour of isolated thin end plate connections up to the ultimate limit state, including end plate-contact and end plate-bolt interaction phenomena. The comparison between computed and measured values in each phase highlights the effectiveness and degree of accuracy of proposed models.

SOMMARIO

Nell'ambito delle attività del progetto di ricerca europeo COST C1 "Civil Engineering Structural Connections", è stata sviluppata una strategia composta da tre fasi che consente la simulazione del comportamento rotazionale di giunti flangiati di acciaio. Nella prima fase, collegamenti a T elementari caratterizzati da diversi meccanismi di collasso plastico sono simulati tramite un modello tridimensionale a elementi finiti con l'impiego del codice di calcolo LAGAMINE. La seconda fase riguarda la calibrazione del codice di calcolo ABAQUS, adottando LAGAMINE come codice di riferimento. Nella terza fase, che è in corso di svolgimento, il codice ABAQUS è impiegato per riprodurre il comportamento elasto-plastico di collegamenti flangiati isolati sottili fino allo stato limite ultimo, includendo sia i fenomeni di contatto che quelli di interazione piastra-bullone. Dal confronto tra i valori di calcolo e quelli sperimentali si può valutare l'efficacia ed il grado di accuratezza dei modelli proposti.

1. INTRODUCTION

Most of present regulation codes regarding constructional steelwork allow for the semi-rigid concept as model for connection behaviour (Eurocode 3, 1992). Numerous examples show the interest of such a modelling in terms of economy (Anderson et al., 1994), but also in terms of safety in the case of seismic action (McMullin et al., 1993). During last years, however, it appeared some reluctance from designers to use this concept, also because of lack of moment-rotation (M - φ) relationships for day-to-day practice. Since connections are generally highly redundant, several investigations were carried out to predict the (M - φ) curve, or at least to approximate the key parts of this relationship in a convenient mathematical format (Bursi, 1994). In this context, the finite element technique represents a rational supplement to those tools and can serve as a basis of design draft. However, it has been long apparent that papers showing excellent agreement between experimental and finite element analysis results merely mean that factors within numerical analyses have been adjusted.

The finite element simulation of bolted connections is generally complicate, because combined non-linear phenomena like friction, slippage, contact and bolt-plate interaction must be reproduced. Latest generation finite element codes can simulate those complex phenomena. However, difficulties remain to the numerical analyst which has to choose appropriate finite element models able to determine an accurate representation of the physics with the lowest computational cost. Choice of number of nodes, shape functions and number of integration points, depend on availability, problem, geometry, type of loading and required accuracy (Zienkiewicz and Taylor, 1991).

Within the Numerical Simulation Working Group of the European research project COST C1 "Civil Engineering Structural Connections" devoted to the development of semi-rigid design procedures, it is intended to use factors for the finite element analysis which are identifiable by reference to explicit guidelines according to the type of loading, finite elements and experimental or characteristic material properties as well. Only when such guidelines have shown to provide reasonably accurate solutions with a wide range of test configurations can numerical predictions of as yet untested components be accepted with confidence. As a result, elementary nonpreloaded and preloaded tee-stub connections tested by Jaspart (1994) and afterwards by Bursi (1995), are used to verify the capabilities of available software packages.

In the first phase of that activity, a 3-D finite element model for elementary tee-stub connections has been set with the LAGAMINE software package. The model includes first-order solid (Li et al., 1992) and contact elements (Habraken et al., 1992), to simulate accurately elementary tee-stub connections up to the ultimate limit state. Then, a calibration phase follows in which based on LAGAMINE simulations, specific elements of the ABAQUS library (Hibbitt et al., 1994b) are chosen. Furthermore, simulations are performed to validate a simplified model, i.e. the spin, which is intended to reproduce the bolt behaviour.

In the third and final phase, not yet completed, the ABAQUS code is used to simulate the elastic-plastic behaviour of end plate connections up to the ultimate limit state, including end plate contact and end plate-bolt interaction phenomena. The numerical results, limited to thin end plate connections, are compared to the experimental ones in terms of generalized displacement and stress distributions, thus assessing the reliability of both models and algorithms.

2. A 3-D FINITE ELEMENT MODEL FOR TEE-STUB CONNECTIONS

At present, complex 3-D finite element analyses of tee-stubs were carried out by Rothert et al. (1992), to idealize the behaviour of end plate connections. A satisfactory agreement between computed and measured values was obtained. However, additional simulations of end plate connections were performed with a 2-D discretization, thus limitating the plastic failure mechanisms that can be effectively captured (Bursi, 1991).

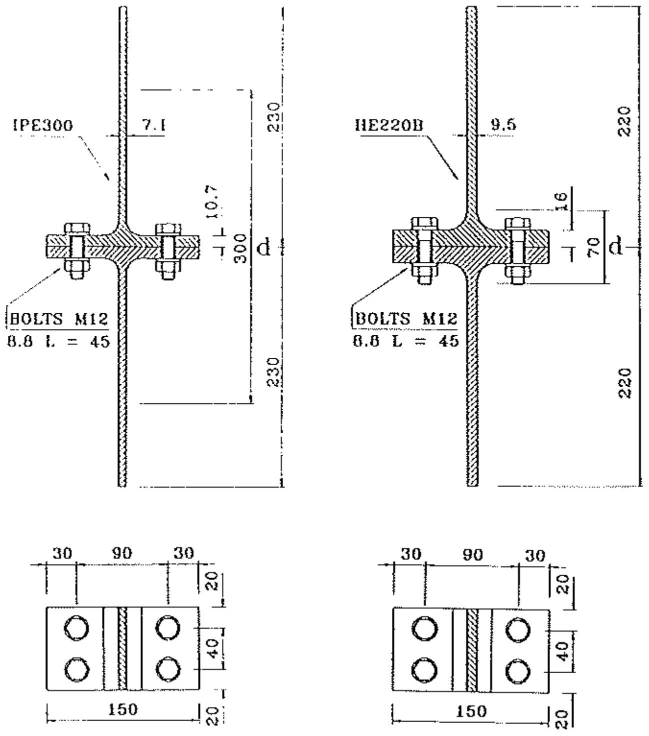
In pursuit the first phase, an elementary tee-stub connection proposed by Jaspart (1994b) and identical to that shown in Fig. 1a is used to test the capabilities of the LAGAMINE software package. This deceptively simple-looking tee-stub specimen deserved some problems in preliminary simulations, because some material data were missing. Hence, additional specimens were tested at the University of Trento under monotonic loading (Bursi, 1995). They are labelled T1 and T2 from now on and are represented with their geometrical characteristics and details in Fig. 1a and 1b, respectively. Specimens were purposely designed to collapse according to the kinematic mechanisms reported in Fig. 2a and 2b, respectively (Eurocode 3, 1993).
Average tensile values of yield stress f_y and ultimate stress f_u of flange and web material are reported in Table 1. Fasteners were M 12 grade 8.8 bolts, characterized by yield and ultimate stresses as shown in Table 2. Within T1 and T2 specimens nonpreloaded and prealoaded bolts were used. In particular, a preloading force $S = 60.7$ kN was applied. In order to perform realistic finite element simulations, material data are reproduced with piecewise linear constitutive laws of the type shown in Fig. 3.

The LAGAMINE finite element software package which has been developed at the MSM Department of the University of Liege is used to perform numerical simulations. Non-linear analyses are carried out according to a Total Lagrangian Formulation which includes all kinematics effects due to large displacements, large rotations and large strains. The objective Jaumman (co-rotational) stress and corresponding strain rate measures are employed in the response description (Cescotto and Habraken, 1991).

The finite element model which reproduces a quarter of the tee-stub connection, incorporates the following LAGAMINE element types.
(*i*) Eight-node first-order hybrid hexaedral elements (Li et al., 1992) intended for use with elastic-plastic material behaviour. For plasticity-type (hyperbolic) problems eight-node elements perform far better than other solid elements because discontinuities in the displacement gradient field are allowed. In addition, the element is based on a mixed formulation with reduced integration and hourglass control, to avoid volumetric and shear "locking" (Li et al., 1992). (*ii*) Contact elements used to reproduce the conditions involving sticking, frictional sliding and flange separation under large displacements (Habraken et al., 1992). The sticking friction is reproduced with a penalty method while a classical isotropic Coulomb friction model is used. The friction coefficient is set to be zero due to symmetry. The Huber-v.Mises-Hencky yield criterion is used to reproduce the ductile yielding of steel. Preloading forces are applied as initial stresses in bolts. A simplifying assumption is made in the washer modelling which is considered jointed to the bolt head. The effect of nut, washer and threaded part of bolt shank are incorporated into an effective shank length according to the bolt model of Agerskow (1976).

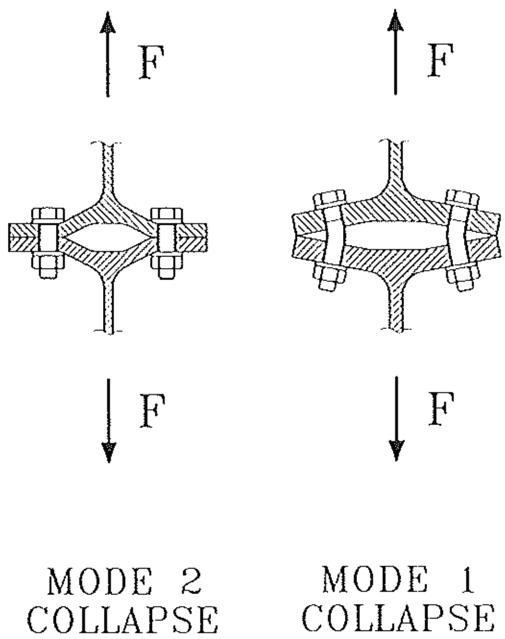
Finite element analyses covered nonpreloaded as well as preloaded specimens. Displacement and related stress field distributions presented here are restricted to prealoaded specimens. The displacement field at the ultimate limit state traced by the finite element analysis is reported in Fig. 4a for the specimen T1. One can observe how the model is able to reproduce the complex kinematics both in the flange and bolt zone. The corresponding distribution of effective stresses is reported in Fig. 4b. The high stress field close to the bolt hole and the flange corner identifies two yield lines which govern the kinematic mechanism observed at collapse. The evolution of effective stresses in each bolt and washer can be observed both in the elastic regime (Fig. 5a) and at collapse (Fig. 5b), respectively. High effective stresses in the bolt shank can be observed. The accuracy of the model can be appreciated by superimposing the computed load-displacement (F - Δd) relationships upon the measured ones, as shown in Fig. 6. The definition of d is found in Fig. 1a. The comparisons are carried out both for nonpreloaded (Fig. 6a) and preloaded (Fig. 6b) specimens and predictions of the finite element model appear quite good. The discrepancies between simulations and tests at the onset of yielding are due to imperfection and residual stress effects which are not taken into account in the model.

The displacement and stress field at collapse for the preloaded tee-stub connection T2, characterized by a flange thickness $t_p = 16$ mm are illustrated in Fig. 7. One can observe how the LAGAMINE



(a) (b)

Figure 1



(a) (b)

Figure 2

Table 1: Characteristics of Tee-Stubs

TEE-STUB	t_p (mm)	Yield stress f_y (MPa)	Tensile stress f_u (MPa)
Flange (T1)	10.7	431	595
Web	7.1	469	591
Flange (T2)	16.0	282	483
Web	9.5	290	498

Table 2: Characteristics of Bolts

BOLT	Φ (mm)	Yield stress $f_{y,b}$ (MPa)	Tensile stress $f_{u,b}$ (MPa)
Shank (T1)	12	893	974
Shank (T2)	12	833	947

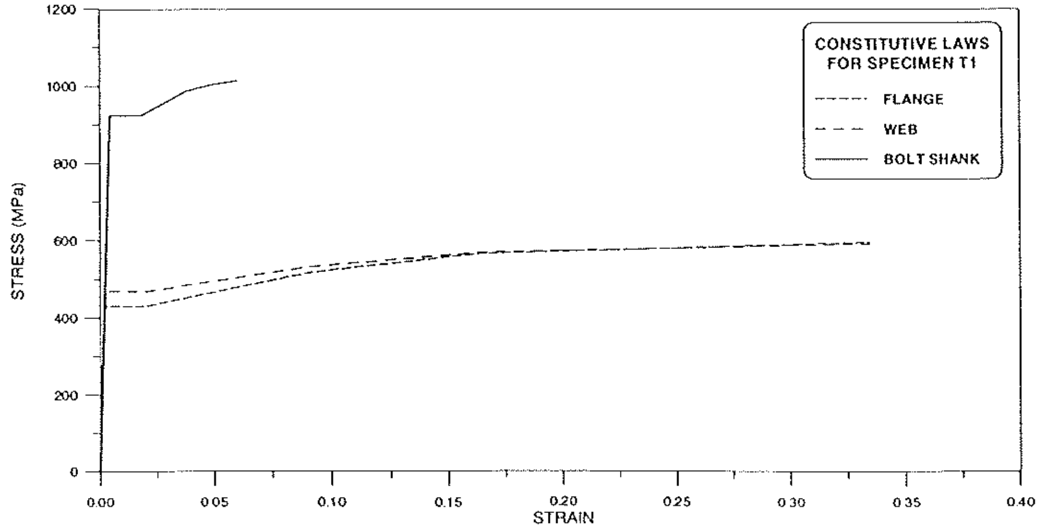
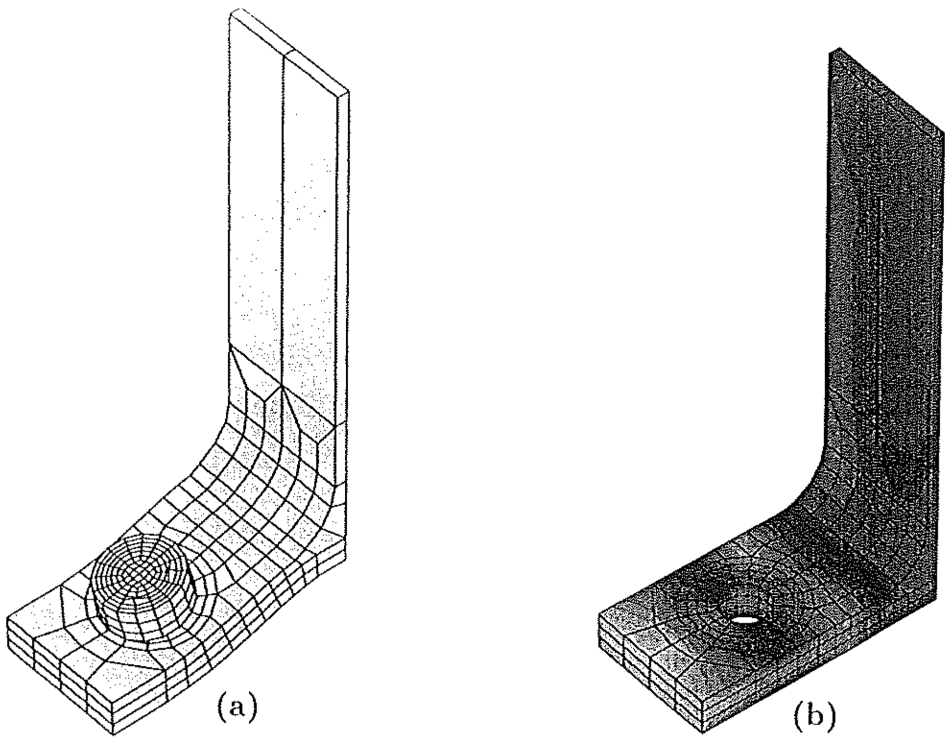


Figure 3

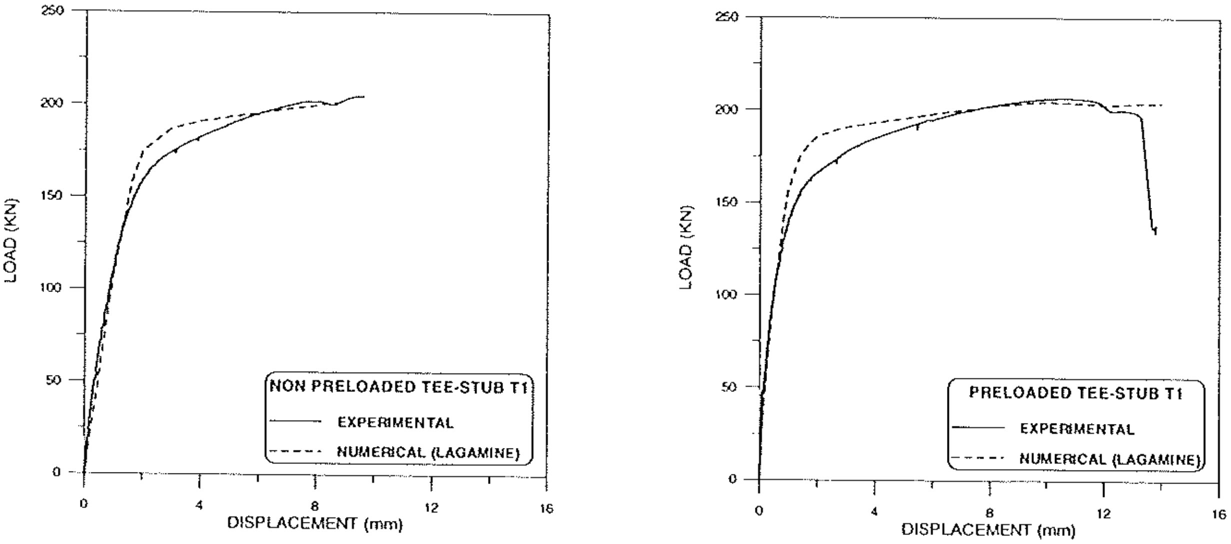


(a) (b)

Figure 4

(a) (b)

Figure 5



(a) Figure 6 (b)

code is able to reproduce a realistic displacement field (Fig. 7a) with the associated kinematic mechanism, characterized by large effective stresses in the tee-stub flange and web (Fig. 7b). The load-displacement (P - Δd) relationships (one may look at Fig. 1b for a proper definition of d) engendered by the model are superimposed upon the corresponding experimental curves in Fig. 8. Even in this case, the displacement evolution is captured with a good accuracy. However, some discrepancies can be observed in strength values.

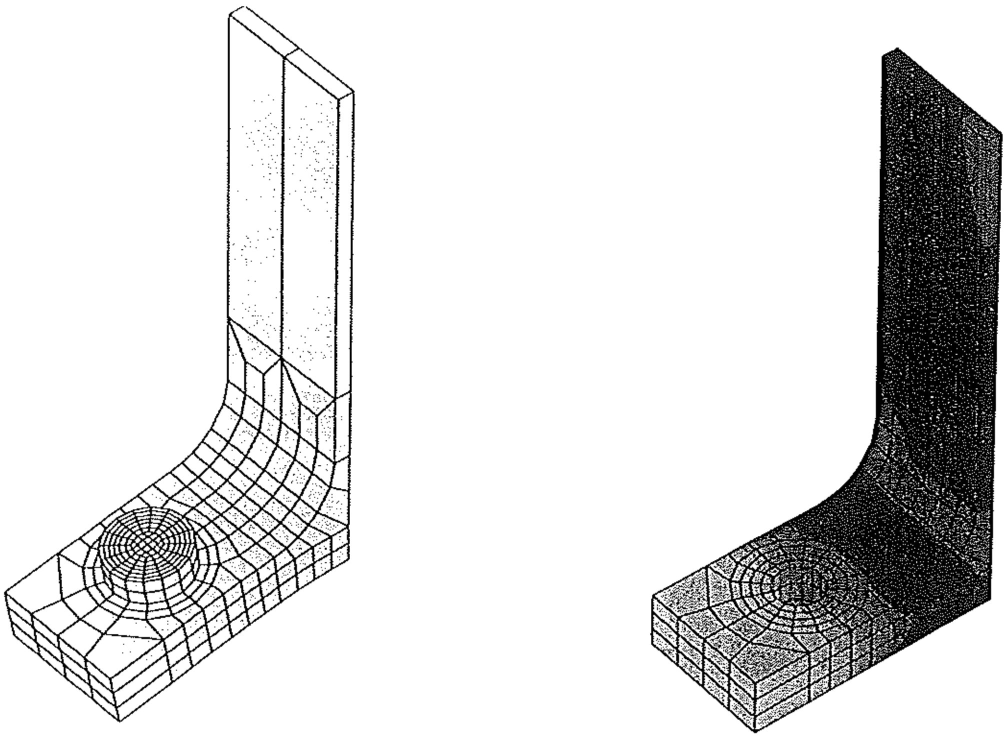
3. CALIBRATION OF CONNECTION MODELS WITH THE ABAQUS SOFTWARE

The software package LAGAMINE is oriented to the solution of sheet forming problems (Zhu, 1993). Therefore, only few elements are available in the code. On the other hand, ABAQUS is a general purpose finite element software package (Hibbitt et al., 1994a) available, nowadays, in many research centres and design offices. Thus, it was decided as second phase, to calibrate finite element tee-stub models generated with the ABAQUS code on LAGAMINE simulations. The software package ABAQUS has a large variety of three-dimensional solid, contact and beam elements (Hibbitt et al., 1994b). Hence, additional simulations were performed on the Tee-stub T1, to be able to discriminate among different elements on the basis of their performances.

In the first set of simulations, the tee-stub T1 without any bolt is examined, as shown in Fig. 9. Boundary conditions constrain the vertical displacement of the the tee flange in the vertical direction around the top part of the bolt head zone and the bottom part of the external edge. As a result, any effect of flange-foundation and bolt-plate interaction is disregarded in the simulations. Thus, only the sensitivity of continuum elements is investigated. The analysis is restricted to 8-node first-order solid elements which are suitable for plasticity-type (hyperbolic) problems, as discussed above. Three elements of the three-dimensional solid element library are examined.
(*i*) Eight-node first-order solid elements with full integration which are characterized by 8 Gauss points per element (C3D8). (*ii*) Eight-node first-order solid elements with reduced integration which have 1 Gauss point per element (C3D8R), to avoid either volumetric or shear locking. (*iii*) Eight-node first-order solid elements with full integration, where the integration is performed on 8 Gauss point per element (C3D8I). However, the element has 13 additional degrees of freedom due to the incompatible modes which can better reproduce any flexural behaviour. The results of these simulations are shown in Fig. 10, where the load-displacement (F - Δd) response is shown, with d defined as in Fig. 1a. If one consider the simulation obtained with LAGAMINE as an "exact response", one can observe that both C3D8 and C3D8I elements perform reasonable well, while C3D8R elements underestimate strength values, as expected. Furthermore, load-displacement responses are characterized by a similar initial stiffness, thus indicating that the number of elements chosen for the mesh is appropriate.

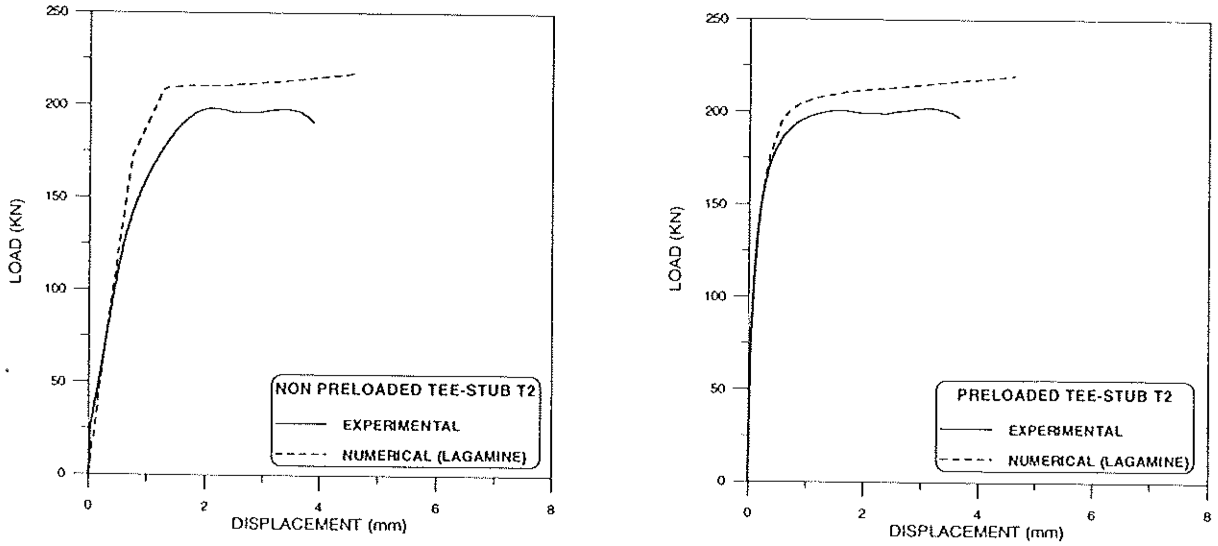
In the second set of simulations, contact elements are added between the foundation and the bottom flange of the tee-stub. In particular, the finite sliding interaction between a deformable and a rigid body is considered in the ABAQUS simulations. Thus, kinematic measures of overclosure and relative shear sliding are used, together with an appropriate Lagrange multiplier technique to introduce contact and friction (Hibbitt et al., 1994a). Separation and sliding of finite amplitude and arbitrary rotations may be reproduced and a Coulomb friction model is used with a value of $\mu = 0$. Results of these simulations are reported in Fig. 11. It can be observed that contact elements perform quite well. As a result, the tee-stub response sensitivity is dictated only by performances of solid elements.

In the ABAQUS code special purpose contact elements which allow for contact between two nodes can be used. Those elements are labelled GAPUNI (Hibbitt et al., 1994b) and allow for two nodes to be in contact (gap closed) or separated (gap open) under large displacements, through a penalty formulation. As a result, their use implies a reduced computational work with respect to contact elements. In addition, a Coulomb friction model and a value of $\mu = 0$ are used to reproduce frictional sliding. Results of simulations carried out to analyze the performances of those simple elements are reported in Fig. 12, where solid elements labelled C3D8 are used both with contact and gap elements.



(a) (b)

Figure 7



(a) (b)

Figure 8

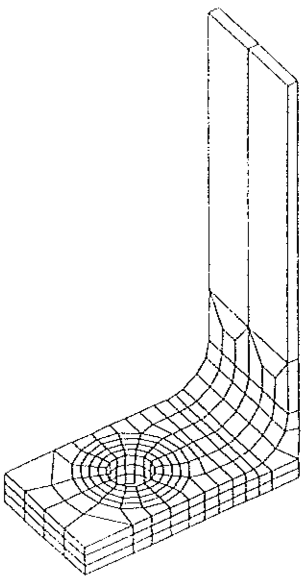


Figure 9

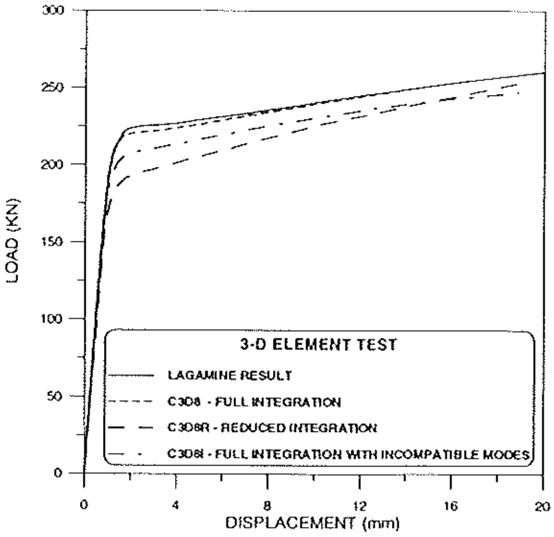


Figure 10

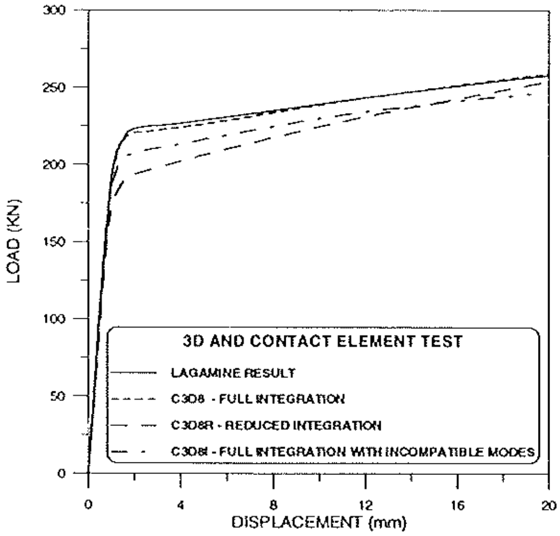


Figure 11

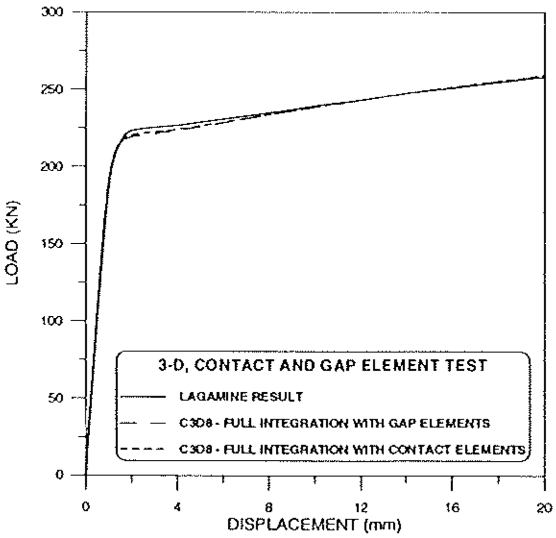


Figure 12

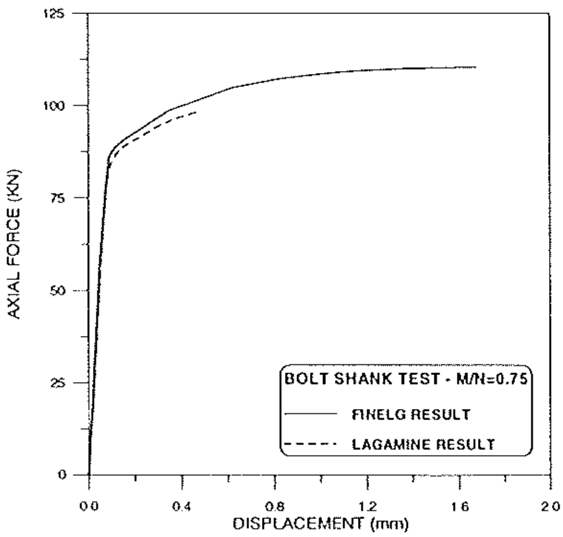


Figure 13

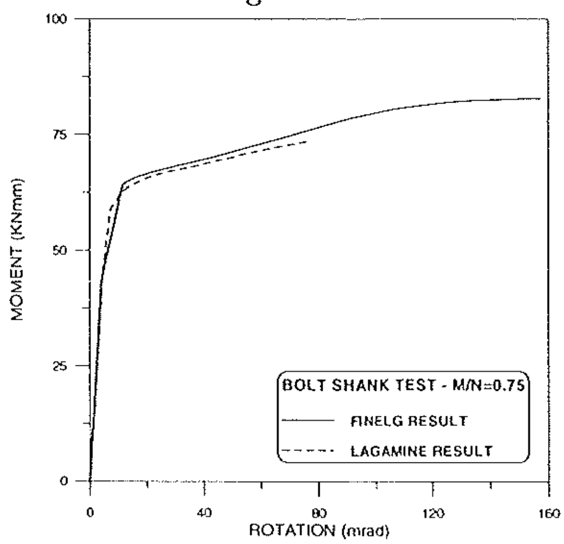


Figure 14

It can be observed how responses with different interface elements are virtually identical.

To avoid a complete 3D modelling of bolts, as done in a previous section, a spin idealization was conceived by Bursi and Leonelli (1994) and by Jaspart (1994a). Hence, additional numerical simulations are carried out, to study the performances of bolt shanks and heads modelled either with 3D solid elements by the code LAGAMINE or with beam elements by using the code FINELG (1995). For conciseness, only simulations related to bolt shanks are presented. Simulations are carried out by subjecting the bolt shank to tensile and flexural stresses thorough the bolt head. In particular, a ratio M/N equal to 0.75 is considered here. The evolution of axial displacement versus the axial force of the shank top is presented in Fig. 13a. One can observe how the beam model reproduced with the FINELG code is able to capture the stiffness and strength behaviour of the bolt shank with a good accuracy. The corresponding moment-rotation $M - \varphi$ relationship is shown in Fig. 13b. Even in this condition, the beam element is able to capture the elastic an inelastic behaviour of the bolt shank. Similar results can be obtained for different M/N loading conditions. With a similar procedure, the solid bolt head was reproduced with a set of beams and verified.

4. A 3-D FINITE ELEMENT MODEL FOR END PLATE CONNECTIONS

The third on-going phase concerns the use a general purpose commercial code, ABAQUS in particular, to reproduce the elastic-plastic behaviour of isolated end plate connections (Bursi, 1991). Similar simulations were performed already by Bursi and Leonelli (1994). However additional assumptions which are removed here, determined only satisfactory results at that time.

In the ABAQUS code, non-linear analyses are performed according to a Total Lagrangian Formulation which includes all kinematics effects due to large displacements, large rotations and large strains. As a result, the Cauchy stress and the rate of deformation measures are employed in the response description. The model, which reproduces a half of an extended end plate connection is set on the basis of the previous calibration phase. As a result, first-order elements C3D8 and C3D8I are used for modelling the continuum. Elements C3D8R are adopted for comparison only. In addition, contact elements are used to reproduce conditions involving sticking, frictional sliding and end plate separation under large displacements. As proved above, GAPUNI elements could be used as well. The Coulomb friction model with a mean value of $\mu = 0.25$ is adopted. Standard two-node beam elements (B31) are used to model bolt shanks and heads according to the spin idealization. The length of the bolt shank is again established with the Agerskov model (1976). Geometric and material characteristics of specimens can be found in Bursi and Leonelli (1994).

Finite element analyses is covering several specimens. However, the output presented here is restricted to the limiting case of a thin (EP 1-1) connection, characterized by an end plate thickness $t_p = 12$ mm. Bolts M20 8.8 are used. In the present mesh discretization also weld beads are modelled with solid elements as shown in Fig. 15a. As a result, a yield stress $f_{y,w}$ and an ultimate stress $f_{u,w}$ equal to 465 and 538 N/mm^2 were assumed for weld beads, respectively. The piecewise linear constitutive model for the weld material is calibrated on the stress-strain law determined by Miazga and Kennedy (1989). The Huber-v.Mises-Hencky yield criterion is used to reproduce ductile yielding of steel. Prestressing forces are applied in the tension bolts as initial stresses, to simulate the snug tight condition. In addition, consistent nodal forces are applied to tension and compression beam flanges to simulate a flexural action. The complete model has 3684 elements and 19600 nodes.

The undeformed mesh relative to half thin end plate EP 1-1 is shown in Fig. 15a. The beam assemblies used to reproduce bolt heads and shanks can be observed. The corresponding displacement field at the ultimate limit state as traced by the analysis is reported in Fig. 15b. The model appears very accurate in reproducing the complex kinematics dominated by bending around the plate tension zone (Bursi, 1991). The moment-rotation (M - φ) relationship determined by the model with different solid elements described above is superimposed upon the corresponding experimental curve in Fig. 16. The model constituted by C3D8I elements is able to approximate with a good accuracy

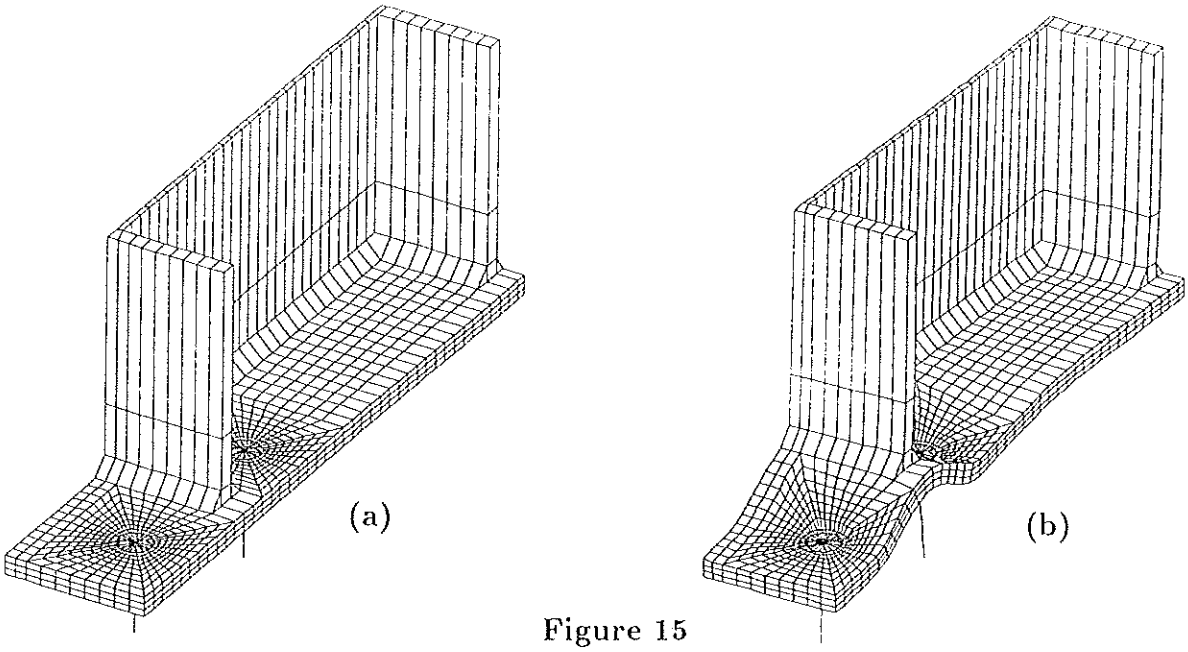


Figure 15

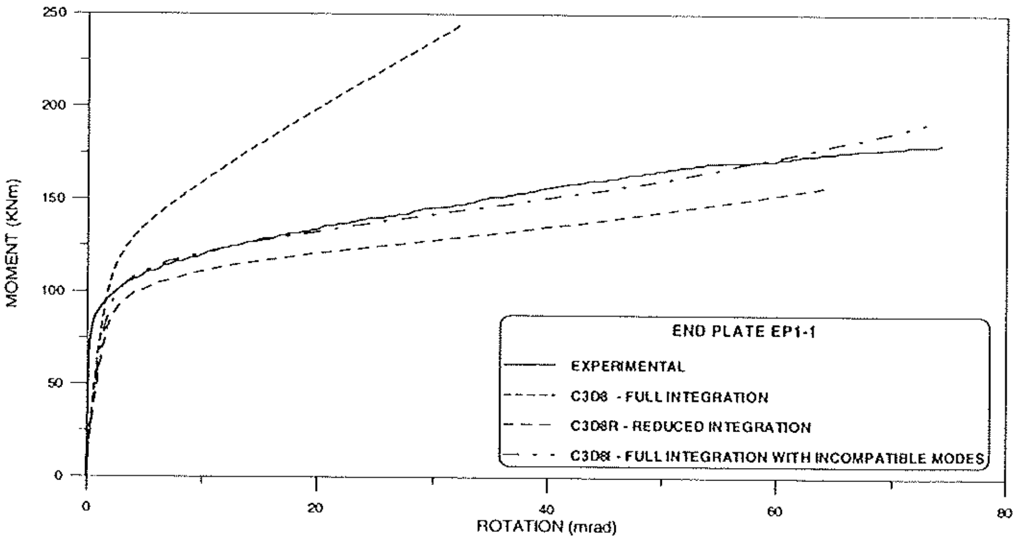


Figure 16

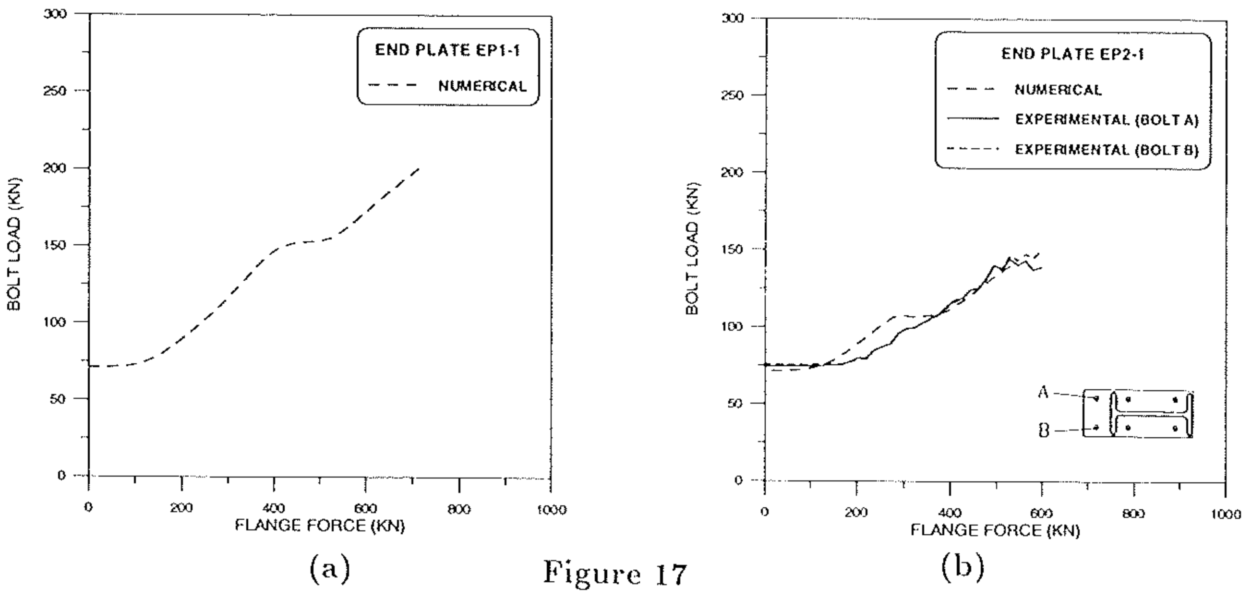


Figure 17

the measured moment-rotation curve, because the thin end plate behaviour is mainly governed by bending. Lack of fit phenomena, imperfections and residual stresses determine some discrepancy in the stiffness behaviour. At the ultimate limit state the simulation is able to reproduce membrane effects which did not happen during testing due to a premature buckling of the compressed beam flange (Bursi, 1991). The simulation performed with C3D8 elements does not appear suitable to simulate the bending behaviour of thin end plates, whereas C3D8R elements underestimate the strength behaviour, as expected. The evolution of the bolt axial force versus the beam flange force $\frac{M}{(h-t_f)}$ is reported in Fig. 17a. The effect of prying forces and end plate yielding can be captured from this type of simulation. Bolt axial forces versus beam flange forces are available by testing for the end plate connection EP 2-1, which has similar geometrical characteristics of end plate EP 1-1 (Bernuzzi and Zandonini, 1990). The comparison between computed and measured values shows a satisfactory agreement in this case. However, the underestimation of end plate stiffness on the onset of yielding determines large values of prying forces in that regime and, as a results, high forces in the bolts.

5. CONCLUDING REMARKS

Within the activities of the COST C1 European research project devoted to the development of semi-rigid design procedures, a three-step strategy that attempts to lead to an accurate simulation of the moment-rotation behavior of extended end plate steel connections has been developed. Relevant phases include the setting of a three-dimensional finite element model with the ABAQUS code, on the basis of tests and simulations performed with the reference code LAGAMINE. The model is able to reproduce the elastic-plastic behaviour of elementary tee-stub connections up to the ultimate limit state. In the final phase, not yet completed, the ABAQUS code is used to reproduce the elastic-plastic behaviour of thin end plate connections up to the ultimate limit state including plate contact and plate-bolt interaction phenomena. The comparison between computed and measured values has highlighted the effectiveness and degree of accuracy which can be reached by those models.

ACKNOWLEDGEMENTS

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

REFERENCES

ABAQUS - Theory Manual. 1994. Version 5.4, Hibbitt, Karlsson & Sorensen Inc.

ABAQUS - User's Manual. 1994. Vol. I & Vol II. Version 5.4, Hibbitt, Karlsson & Sorensen Inc.

Agerskov, H. 1976. High-Strength Bolted Connections Subject to Prying. *Journal of Structural Engineering Division, ASCE* 102-1:161-175.

Anderson, D., Colson, A. & Jaspart, J.P. 1994. Calcul D'Ossatures et Assemblages: Considerations Economiques. *Construction Metalliques* 1:45-52

Bernuzzi, C. & Zandonini, R. 1990. Moment-Rotation Response of Extended End Plate Connections. *Proc. of Int. Colloquium on Stability of Steel Structures, Budapest:* 155-162.

Bursi, O.S. 1991. An Experimental-Numerical Method for the Modelling of Plastic Failure Mechanisms of Extended End plate Steel Connections. *Structural Engineering Review* 3:111-119.

Bursi, O.S. 1994. Design of Semi-rigid Joints in Steel Frames - Part 3 - Annex B: Methods for Joint Characterization. *European Convention for Constructional Steelwork, Technical Work Group 10.2, Brussels*, draft.

Bursi, O.S. & Leonelli, L. 1994. A Finite Element Model for the Rotational Behavior of End Plate Steel Connections. *Proc. 50th Anniversary Conference, SSRC, Chicago.*

Bursi O.S. 1995. A Refine Finite Element Model for Tee-stub Steel Connections, *COST C1, Numerical Simulation Working Group.* Doc. C1WD6/95-07.

Cescotto, S. & Habraken, A.M. 1991. A Note on the Response to Simple Shear of Elasto-Plastic Material with Isotropic Hardening. *Euro. J. of Mech., A/Solids.* 10:1-13

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

Eurocode 3 - Annex JJ: Joints in Building Frames. 1993. *CEN/TC250/SC3/PT9/Part 1b.*

FINELG - Non Linear Fem Computer Program. 1995 *MSM Department, Institute du Genie Civil, University of Liege.*

Jaspart, J.P. 1994a. Numerical Simulations of Joint Behaviour at the University of Liege. *COST C1, Numerical Simulation Working Group*, Doc. C1WD6/94.

Jaspart J.P. 1994b. Numerical Simulation of a T-stub - Experimental Data. *COST C1, Numerical Simulation Working Group.* Doc. C1WD6/94-09.

Habraken, A.M., Radu, J.P. & Charlier R. 1992. Numerical Approach of Contact with Friction Between Two Bodies in Large Deformations. *Proc. of Contact Mechanics Int. Symposium, Lausanne.* 1992.

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

McMullin, K.M., Astaneh-Asl, A., Fenves, G.L. & Fukuzawa, E. 1993. Innovative Semi-Rigid Steel Frames for Control of the Seismic Response of Buildings. *Report No., UCB/EERC-93/03, University of California,* Berkeley. CA.

Miazga, G.S. & Kennedy, D.J.L. 1989. Behaviour of Fillet Welds as a Function of the Angle of Loading. *Canadian. J. Civil Eng.* 16:583-599.

Rothert, H., Gebbeken, N. & Binder B. 1992. Non-Linear Three-Dimensional Finite Element Contact Analysis of Bolted Connections in Steel Frames. *Int. J. for Num. Meth. in Eng.* 34:303-319.

Zienkiewicz, O.C. & Taylor, R.L. 1991. *The Finite Element Method.* 1, Fourth Edition, McGraw-Hill.

Zhu, Y. 1993. Contribution to the Local Approach of Fracture in Solid Dynamics. *Ph. D. Thesis, MSM Department University of Liege.*