Multi-Scale Modelling of Flexible End Plate Connections under Fire Conditions

Ying Hu*, Buick Davison, Ian Burgess and Roger Plank

Department of Civil and Structural Engineering, The University of Sheffield, Sheffield S1 3JD, UK; School of Architecture, The University of Sheffield, Sheffield S10 2TN, UK

School of Architecture, The University of Sheffield, Sheffield S10 2TN, UK

Abstract: Conducting experimental tests is an attractive and straight-forward research approach but is time-consuming and expensive in comparison with finite element modelling. A numerical approach has been developed in this project to investigate the performance of simple steel connections in fire conditions. This paper presents a quasi-static numerical analysis with cohesive elements to investigate the resistance and ductility (rotation capacity) of simple steel connections (flexible end plates) in fire conditions. In comparison with experimental test data, a good correlation with the finite element analysis is achieved and the method is suitable to study the tying resistance and ductility for simple steel connections with various dimensions at different temperatures.

This numerical approach was also compared with component-based connection models, which have been developed in the previous research work. The analytical results produced demonstrated that the component-based approach is capable of as an alternative method to analyse the connection performance under fire and non-fire conditions, and this approach is simple but without loss of accuracy.

Keywords: Flexible end-plate connections, finite element modelling, explicit, structural fire engineering, fire.

1. INTRODUCTION

Conducting fire tests on structures or isolated structural members is time-consuming, expensive, and poses the additional difficulties of recording movement and strain within a furnace. So the development of accurate predictive methods to simulate the behaviour of steel structures in fire has long been regarded as desirable [1]. The ABAQUS program has been used to study the behaviour of steel and composite framed structures in fire by Corus Research [2, 3] and Edinburgh University [4, 5]. This finite element programme shows the ability to simulate complex structural behaviour under fire conditions as comparisons with test results from the Cardington fire tests demonstrated. This commercial programme has a large library of finite elements to enable efficient and detailed modelling of many of the special features of structural behaviour in fire. The complex real phenomena such as local/distortional buckling, lateral torsional buckling, detailed connection modelling and membrane actions can be accounted for in a numerical fashion. Therefore, detailed finite element modelling of connections in fire provides a good opportunity for wider parametric investigations and eliminates the limitations associated with experiments [6], provided that the numerical models have been adequately validated against experimental data.

Initial attempts to simulate steel connections started with two dimensional models, owing to the limitations in computational resources both in terms of software and hardware. In a 2D model, each component of a connection can be represented by using shell or truss elements, and the interactions between these components are numerically simplified to avoid convergence difficulties in the numerical computation. Because of the rapid improvement in hardware and software, computers are now able to perform more detailed simulations for connections in 3D models. Krishnamurthy *et al.* [7] and Kukreti *et al.* [8] compared numerical results produced by two-dimensional and three-dimensional simulations, and found the three-dimensional numerical model to be more flexible than the two-dimensional counterpart, resulting in larger displacements and stresses. Vegte *et al.* [9] believe that, since bolted steel connections are three-dimensional in nature, two-dimensional numerical models are therefore unable to represent the three-dimensional behaviour satisfactorily. Hence, a three-dimensional non-linear finite element analysis approach has been developed as an alternative method for the investigation of connection robustness in fire.

2. THE FINITE ELEMENT MODEL DESCRIPTION

Sherbourne and Bahaari [10, 11] developed a three-dimensional finite element model for simulating endplate connections by using brick elements. The model was assumed to have a continuous connection between the nodes of the bolt head and nut, and the nodes of end plates, and as a consequence, the relative motions between bolt, column flange and end plates were numerically simplified. The bolt shank behaviour was represented using truss elements instead of brick elements which prevents the numerical model reproducing properly the bearing action between bolts and bolt holes, because the interface between the bolt shank and

---

*Address correspondence to this author at the Department of Civil and Structural Engineering, The University of Sheffield, Sheffield S1 3JD, UK; Tel: 0114 22 25763; Fax: 0114 27 98276;
E-mail: cip05yh@sheffield.ac.uk

$$\Delta t_{stable} = \frac{L^e}{c_d} \tag{1}$$

Where L^e is the element length and c_d is the wave speed of the material, which can be determined by:

$$c_d = \sqrt{\frac{E}{\rho}} \tag{2}$$

Where E is the Young's modulus and ρ is the mass density. The stiffer the material, the higher the wave speed, resulting in a smaller stability limit; the higher the density, the lower the wave speed, resulting in a larger stability limit. For more details of the stability limit, readers are referred to the Abaqus Help Documentation 6.7 [14], which explains an analytical approach for the calculation of the exact stability limit, based on the highest frequency and critical damping in a system.

2.2. Element Types

ABAQUS contains a large variety of hexahedron (brick), shell, contact and beam elements endowed with different features depending on the application. Kukreti *et al.* [8] and Gebbeken *et al.* [15] carried out a comparative investigation on numerical techniques in analyzing bolted steel connections with the intention of reproducing the experimental results in a finite element fashion. They set up a two-dimensional finite element model (using shell elements) and a three-dimensional finite element model (using brick elements) within ABAQUS. The comparison between numerical results and experimental data illustrated that the two-dimensional model was too stiff for the representation of the real deformations [15], and the hexahedron (brick) element was much more suitable to model the continuum behaviour of bolted connections compared to standard shell elements.

The current ABAQUS element library offers engineers and numerical analysts a number of hexahedron elements in finite element simulations. For hyperbolic problems (plasticity-type problems), Bursi and Jaspart [16] suggest that the first order elements are likely to be the most successful in reproducing yield lines and strain field discontinuity. This is because some components of the displacement solution can be discontinuous at element edges. Simulations performed by Bursi and Jaspart [16] compared three eight-node brick elements: (1) The C3D8 element with full integration (8 Gauss points). This element is accurate in the constitutive law integration but the shear locking phenomenon is commonly associated with it when simulating bending-dominated structures [14]. (2) The C3D8R element with reduced integration (1 Gauss point). This element supplies a remedy for the shear locking problem caused by using C3D8, but the rank-deficiency of the stiffness matrix may produce spurious singular (hourglassing) modes [14], which can often make the elements unusable. In order to control the hourglass modes in elements, Flanagan and Belytschko [17] proposed the artificial stiffness method and the artificial damping method in the ABAQUS code; although the artificial damping approach is available only for the solid and membrane elements in ABAQUS Explicit. (3) The C3D8I element with full integration (8 Gauss points) and incompatible modes. This element has 13 additional degrees of freedom and the primary effect of these degrees of freedom is to eliminate the so-called parasitic shear stresses that are observed in regular displacement elements in analyzing bending-dominated problems [14]. In addition, these degrees of freedom are also able to eliminate artificial stiffening due to Poisson's effect in bending.

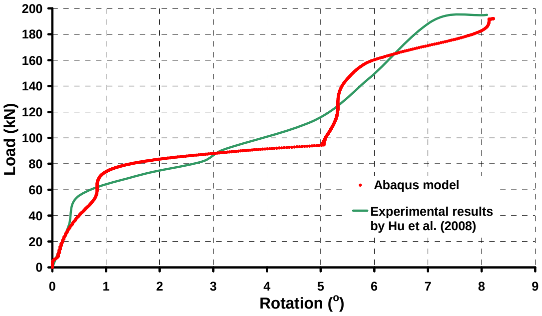
Through comparative modelling with the aforementioned three brick elements, the C3D8I elements were found to perform particularly well both in the elastic and inelastic regimes, and are suitable for representing the bending-dominated behaviour of a structure [12]. As expected from the theoretical formulation, C3D8R elements underestimate the strength value and the plastic failure load in the finite element modelling. From calibration tests, Bursi and Jaspart [12] also state that C3D8 elements appear to be unsatisfactory, owing to the overestimation of the plastic failure load and the shear locking phenomenon. Therefore, in order to predict the behaviour in a conservative fashion, the element selected for bolted steel connections is the reduced integration brick element C3D8R. In order to control the hourglass modes, a very dense mesh finite element model has been set up.

2.3. Contact Modelling Within ABAQUS

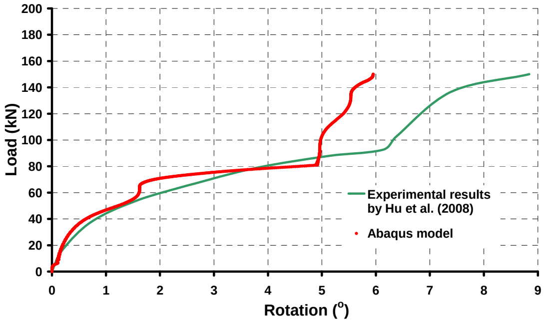
In numerical simulations, obtaining realistic representation of connection performance depends upon handling the difficult issues of modelling the contact interaction between various joint components. Within ABAQUS, the contact behaviour can be simply reproduced by using so-called "gap elements", which require the user to define pairs of nodes and specify the value of a clearance gap. These elements allow for two nodes to be in contact (gap closed) or separated (gap open) under large displacements [12]. The limitation of this sort of element is the friction between two contacted components being ignored in the simulation. Furthermore, simulation using these elements is a tedious and time-consuming task [13].

In order to overcome these problems, a "surface-to-surface" contact interaction was developed for the numerical model. The simulation requires the researcher to first determine the slave and master surfaces for two deformable bodies and then define the interaction behaviour between these two surfaces. In the standard analysis, ABAQUS affords two formulations, small-sliding formulation and finite-sliding formulation, for modelling the interaction between two discrete deformable bodies. In the explicit analysis, the interactions between surfaces are modelled by a different contact formulation, which includes the constraint enforcement method, the contact surface weighting, the tracking approach and the sliding formulation. In the explicit analysis, the friction conditions (sliding and sticking) between the master and slave surfaces may be represented by the classical isotropic Coulomb friction model, which has proved to be suitable for steel elements [18]. However, it is of great importance to be careful with the assignment of the slave and master surfaces [6]. It is generally accepted that the surfaces working as master surfaces should belong to the bodies with the stronger material or a finer mesh. So the contact surfaces of the bolt shank, bolt head and bolt nut are always modelled as master surfaces in this research, as highlighted in red in Fig. (**4**). In

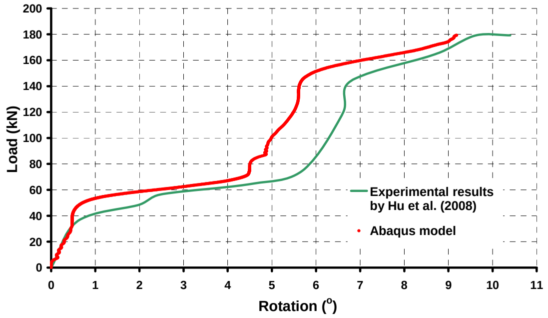
Fig. (9). FE model of flexible end plate connection: deformed and un-deformed shapes.



(a) 35^o



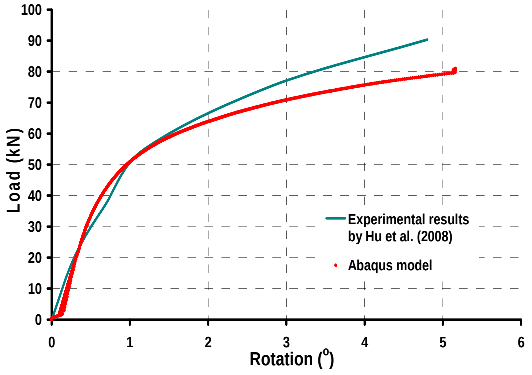
(b) 45^o



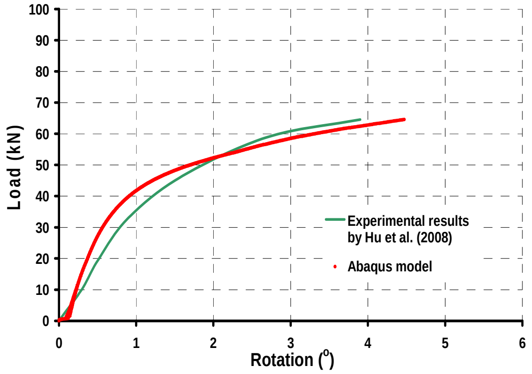
(c) 55^o

Fig. (10). Load-Rotation comparisons between FE model and experimental results for flexible end plate connections (35°, 45° and 55° representing the loading angles of applied forces).

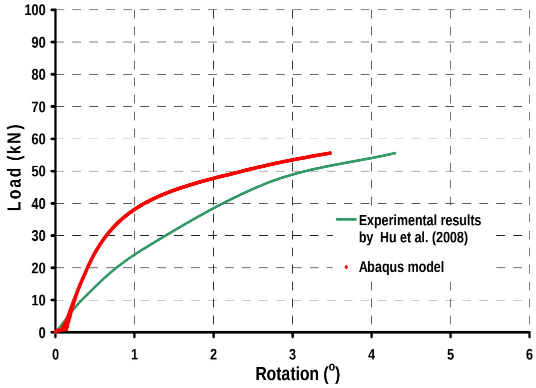
tacted with the column flange. It is apparent in Fig. (**10**) (**a**) and (**c**) that the numerical plots are in good agreement with the experimental plots. Fig. (**10**) (**b**) shows some discrepancy between numerical simulation and experimental results, which might be caused by slight variation in geometrical and material properties between structural components such as bolts, welds and endplates in the specimens tested, which is unable to be represented by a unique homogeneous and isotropic finite element model.



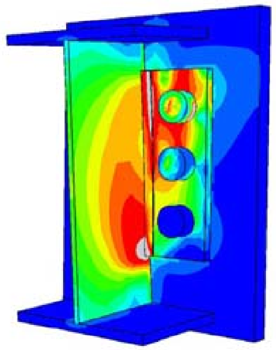
(a) 35^{o} - 450 ^{o}C



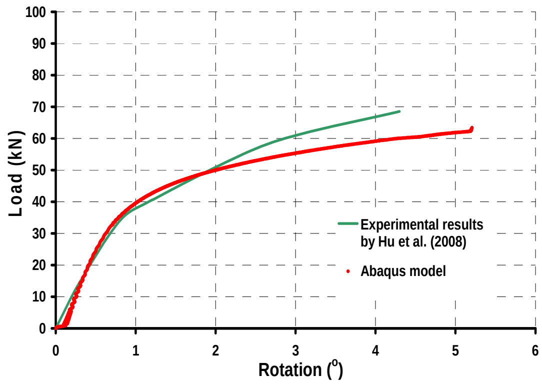
(b) 45^{o} - 450 ^{o}C



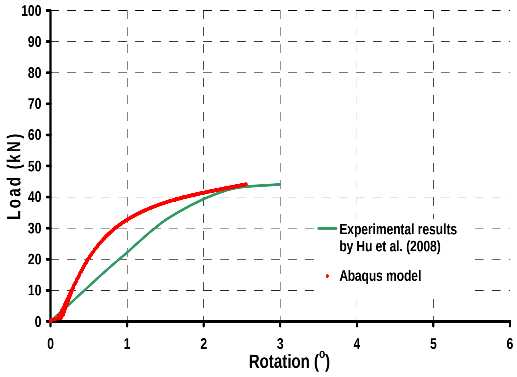
(c) 55^{o} - 450 ^{o}C



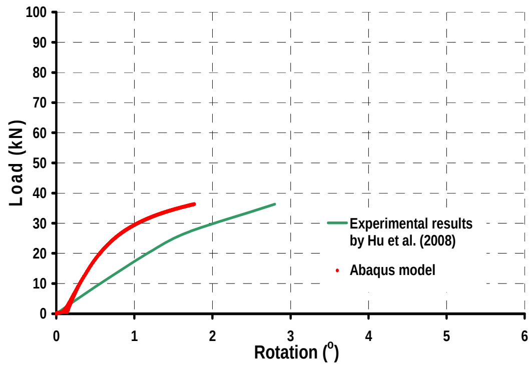
(d) Deformed shape at 450 ^{o}C



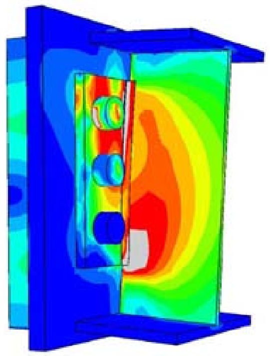
(e) 35^{o} - 550 ^{o}C



(f) 45^{o} - 550 ^{o}C

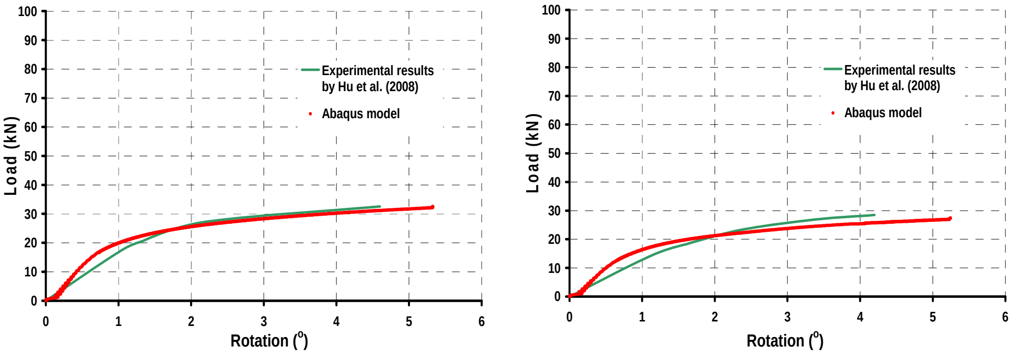


(g) 55^{o} - 550 ^{o}C



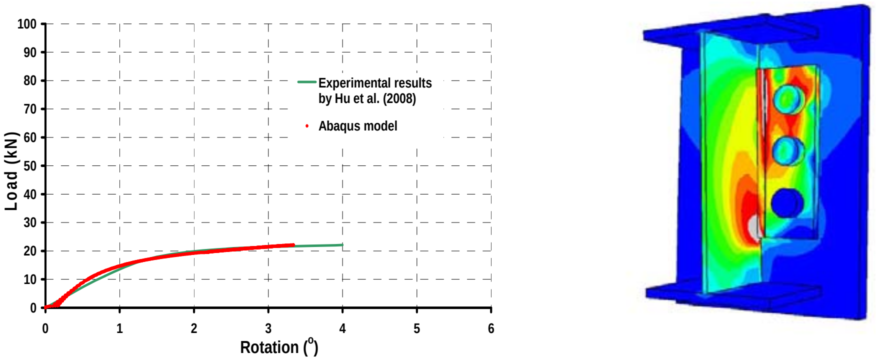
(h) Deformed shape at 550 ^{o}C

Fig. (11). Contd….



(i) 35° - 650 °C

(j) 45° - 650 °C



(k) 35° - 650 °C

(l) Deformed shape at 650 °C

Fig. (11). Load-Rotation comparisons between FE model and experimental results for flexible end plate connections (35°, 45° and 55° representing the loading angles of applied forces).

3.2. Comparison of Flexible End Plate Model at Elevated Temperatures

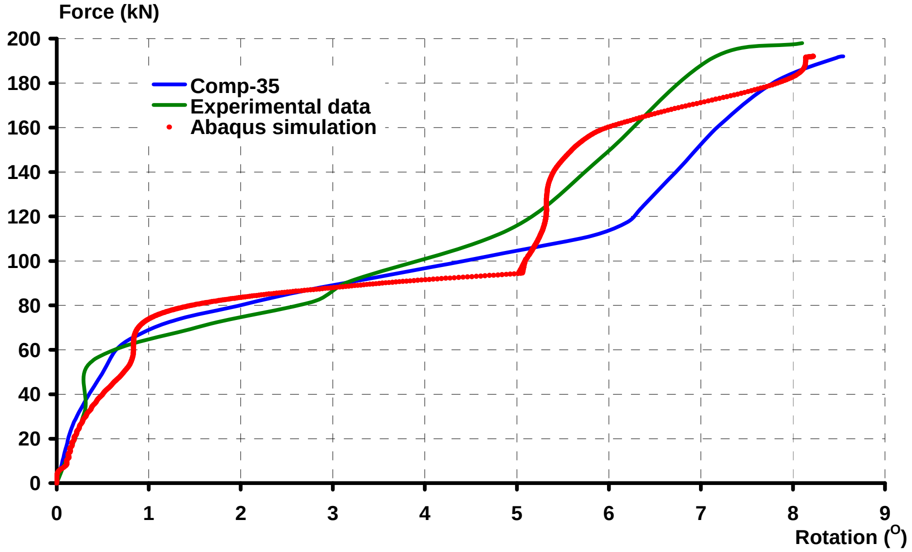
Nine experimental tests for flexible end plates have been carried out at temperatures of 450 °C , 550 °C and 650 °C, and the relationships of loads versus rotations are plotted in Fig. (**11**). A series of finite element simulations have also been conducted and compared with experimental results. The numerical model requires the material properties of steel to be applied at the predetermined temperatures, and the reduction retention factors used are based on recommendations from EC3 [27]. In the finite element modelling, thermal conduction is assumed to have no effect on the connection performance at very high temperatures because of uniform temperature distribution in the specimens. The deformed shapes of connections and finite element models are also shown in Fig. (**11**) for each temperature.

The experimental and numerical plots demonstrate that the complex finite element models are able to predict the connection behaviour at both ambient and elevated temperatures. It was also observed that the connections, both in the numerical simulations and the experiments, failed by the rupture of endplates before the beam flange contacted with the column flange due to the high stress concentration around the heated affected zone (HAZ). Except for Fig. (**11**) (**g**), the curves of loads and rotations, produced by numerical simulation, are in good agreement with recorded experimental data. However, some discrepancies between the numerical analysis and experimental data should be accepted in the finite element simulation, which is because the unique homogeneous and isotropic finite element model are unable to represent the variability in structural components such as material properties and geometrical features, especially under fire conditions.

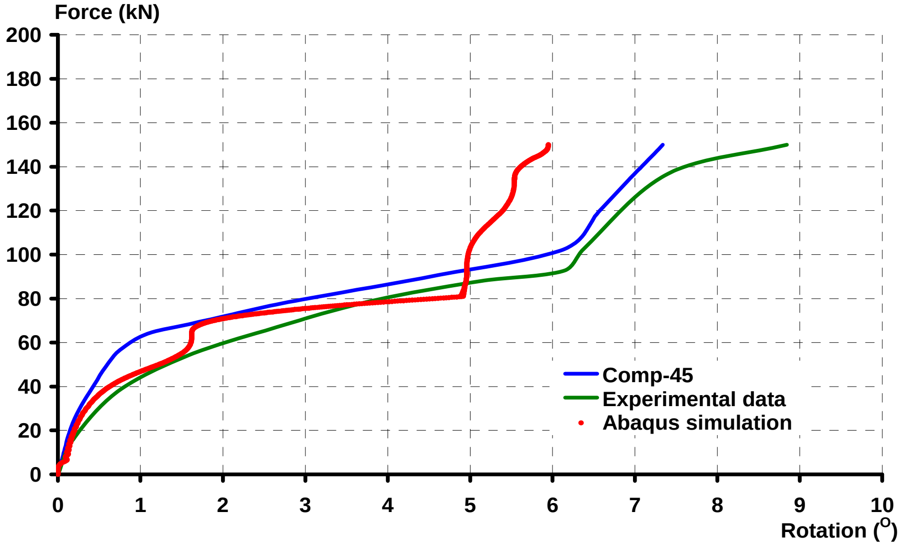
4. THE COMPONENT-BASED MODEL (SIMPLIFIED MODELLING)

The component-based connection model, as explained in the journal publication of Hu *et al*. [28], has been used as a simplified approach to represent the connection behaviour. The basic procedure in application of this approach requires the following three steps [29, 30]:

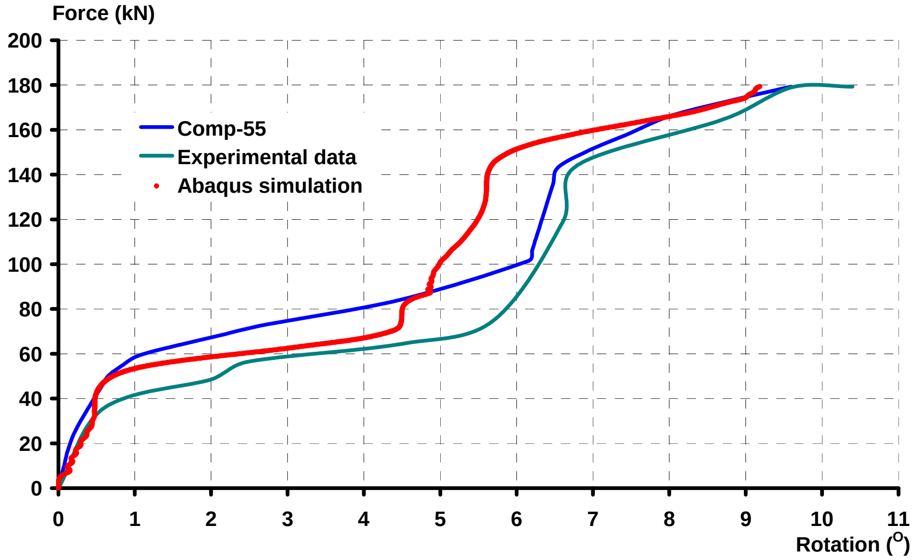
i) Identification of the active components in a structural connection. The active components for an endplate connection have contained tension zone components (bolts, welds and T-stubs), compression zone components and vertical shear components (plate bearing component and bolt in single shear). Bolts and welds are classified as brittle elements in a steel connection, and at elevated temperatures, reduction in strength resistance of these components had a side effect on failure mechanisms and ductility of these endplate connections, which has been documented in detail in the journal paper of Hu *et al*. [28]. This is an important reason to have them included in this simplified approach to precisely simulate the fire performance of endplate connections.


Force (kN)
200
180
160
140
120
100
80
60
40
20
0
0
1
2
3
4
5
6
7
8
9
Rotation (^{O})
Comp-35
Experimental data
Abaqus simulation

(a) 35°

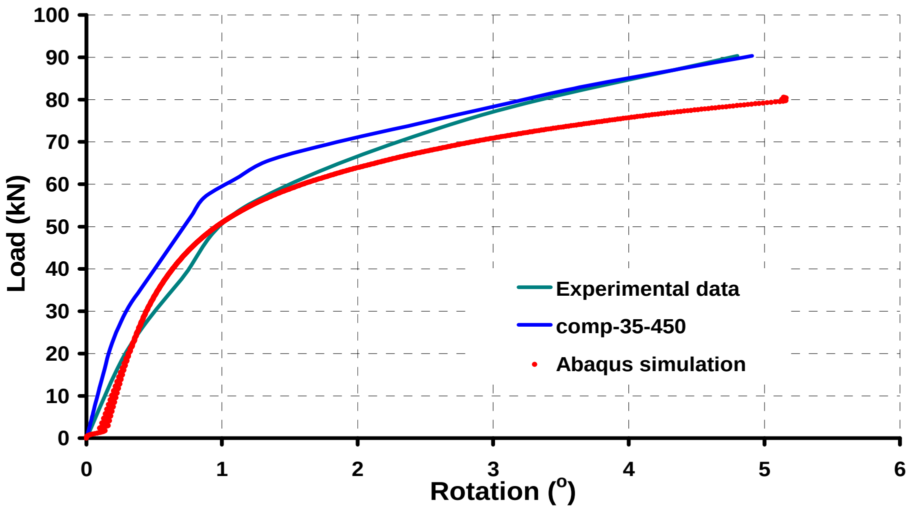

Force (kN)
200
180
160
140
120
100
80
60
40
20
0
0
1
2
3
4
5
6
7
8
9
10
Rotation (^{O})
Comp-45
Experimental data
Abaqus simulation

(b) 45°

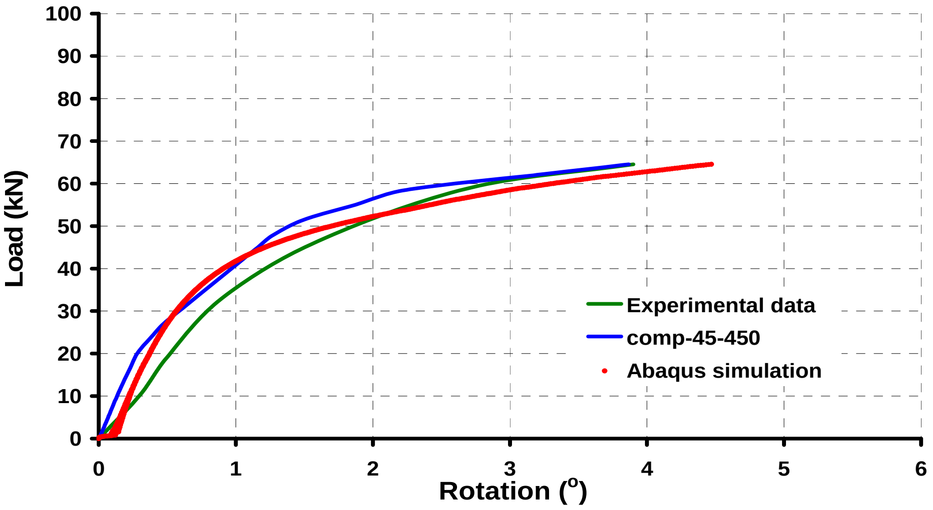

Force (kN)
200
180
160
140
120
100
80
60
40
20
0
0
1
2
3
4
5
6
7
8
9
10
11
Rotation (^{O})
Comp-55
Experimental data
Abaqus simulation

(c) 55°

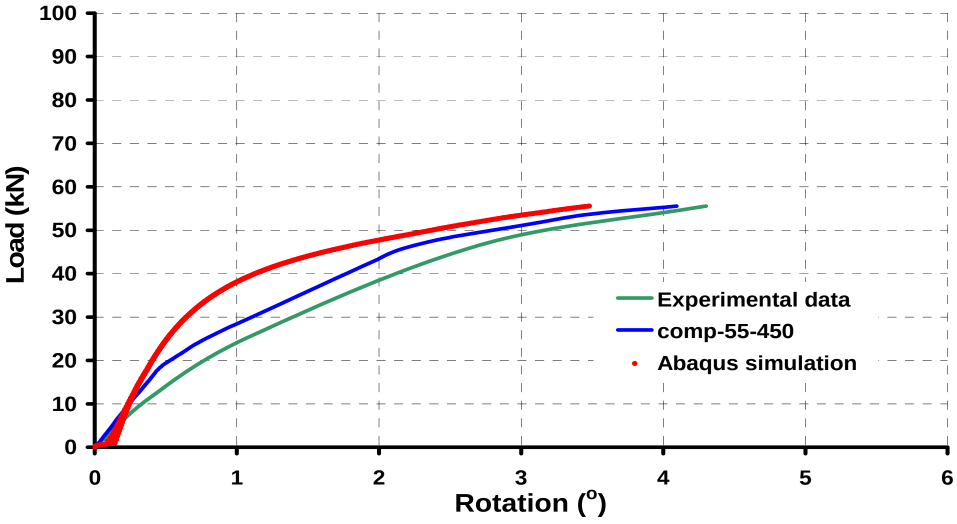
Fig. (12)….



(d) 35° - 450°C

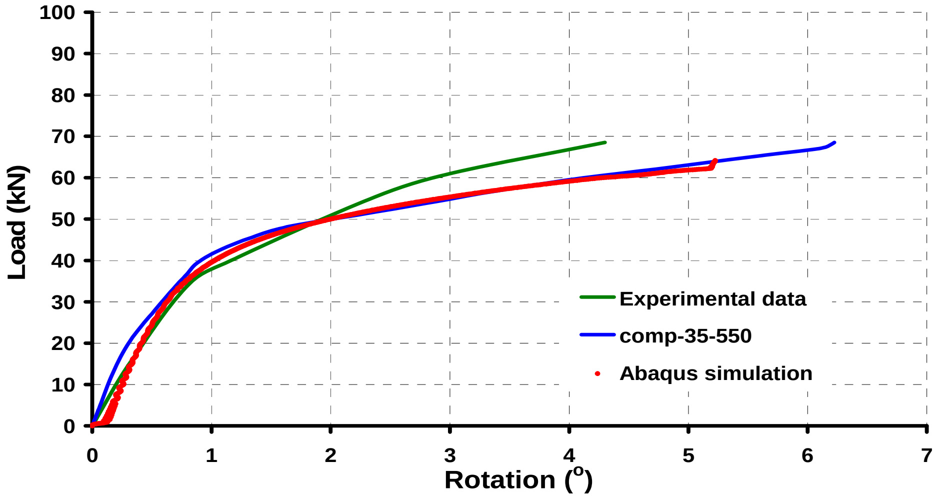


(e) 45° - 450°C

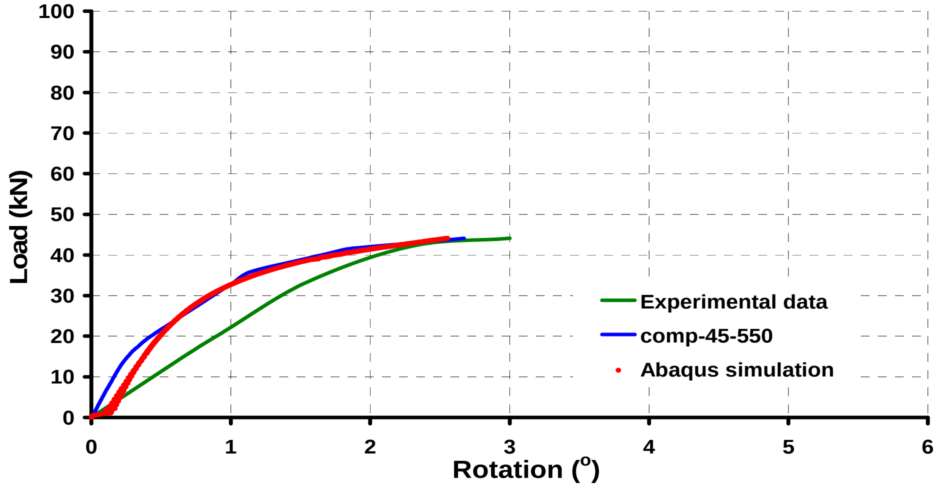


(f) 55° - 450°C

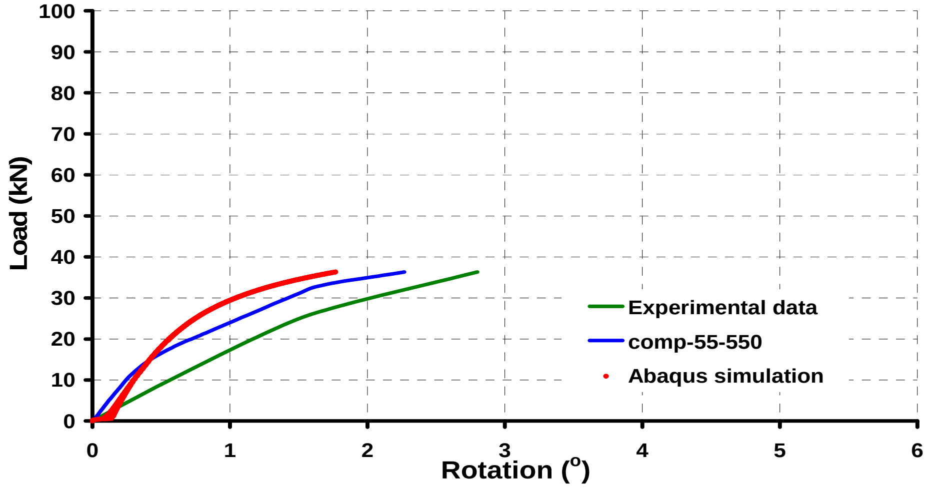
Fig. (12). Contd….



(g) 35° - 550°C

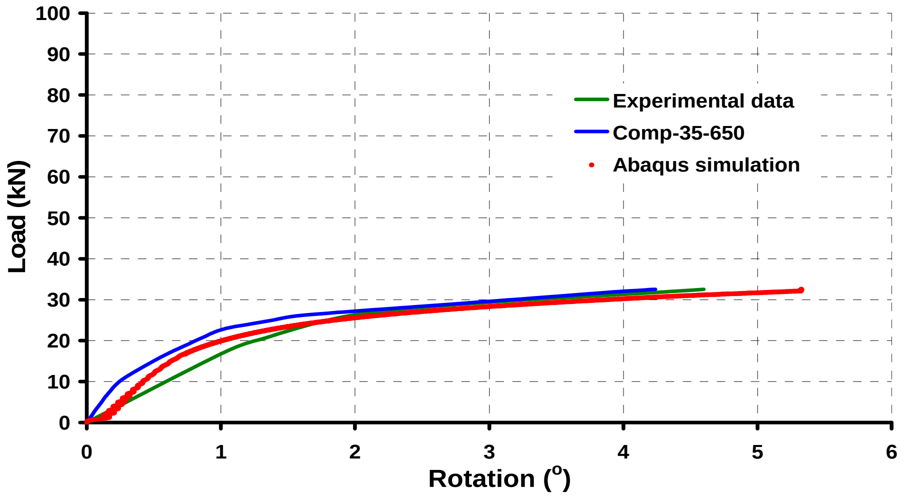


(h) 45° - 550°C

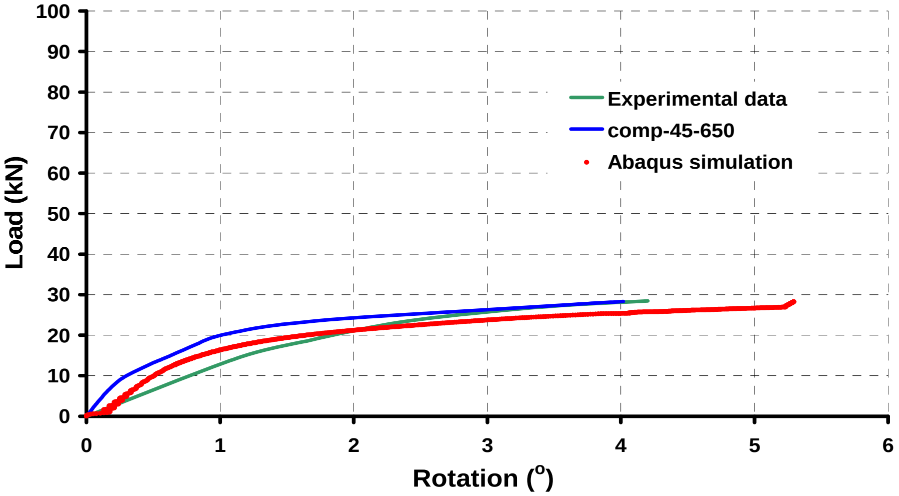


(i) 55° - 550°C

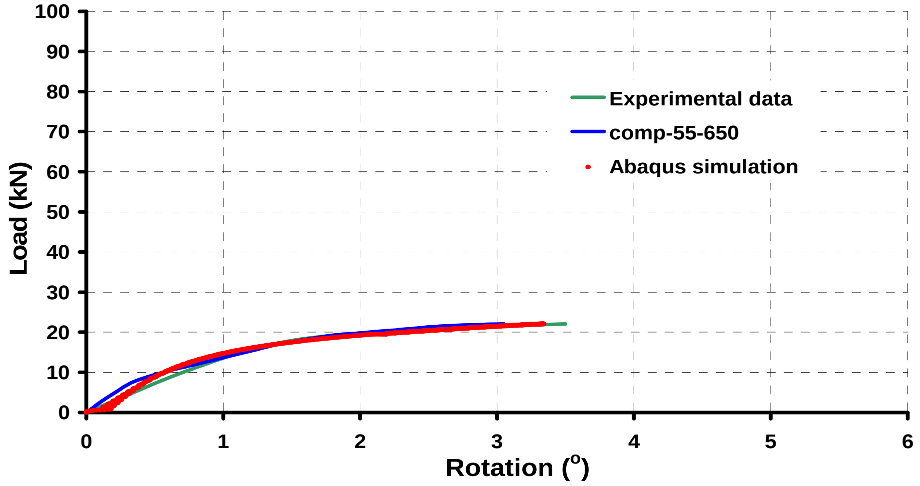
Fig. (12)….



(j) 35° - 650°C



(k) 45° - 650°C



(l) 55° - 650°C

Fig. (12). Comparisons between FE models and component-based models for flexible end plate connections (35°, 45° and 55° representing the loading angles of applied forces).

ii) Characterization of the nonlinear load-displacement response for each individual component

iii) Assembly of all the components and evaluation of the structural response of the whole joint

In order to provide an accurate and practical prediction for steel connection characteristics, this approach has been extended to include the shear components in the vertical direction instead of assuming infinite vertical stiffness. The second stage behaviour of partial depth endplate connections has been simulated with two spring-like components in the compression zone, which enable this model to capture the connection response after the beam contacts with the column surface. Moreover, the effect of brittle components (welds and bolts) has also been taken into account instead of assuming these components as the strongest in these connections; and the experimental results also proved that these components were vulnerable in fire attacks. In the previous section, the finite element model demonstrated a good agreement with the experimental tests. However, utilization of the finite element model (brick elements) to investigate the connection performance is computationally time-consuming for engineering design. Compared to complex finite element analysis, the analytical component-based model is a more simplified and economical approach for design. It is therefore worth conducting a comparison between the numerical analyses and the analytical component-based models, see Fig. (**12**).

In the above plots, the green solid curves represent the relationships of loads and rotations reported in the experimental tests and the blue curves are the connection responses predicted by using the analytical component-based approach. The red dotted curves are from the finite element simulations. The above comparisons demonstrate the analytical component-based model to be an accurate and practical prediction method for steel connection characteristics at both ambient and elevated temperatures. In comparison with the complicated finite element simulation, the component-based model is a more simplified and economical approach without significant loss of accuracy, which is able to aid engineers to identify the weakest structural components under fire conditions. Component-based models are an effective solution for the complex connection behaviour prediction in the design of steel or composite-steel frames.

5. CONCLUSIONS AND RECOMMENDATIONS

This paper reported the development of finite element models embedded with cohesive elements for prediction of the performance of partial depth endplate connections under fire conditions. Comparisons between the finite element simulation and the component-based modelling were presented, which demonstrated the simplified connection model to be capable of representing the complex behaviour of steel joints. This new simplified method may also be used to analyze the performance of a Rugby-post sub-frame under fire conditions. In the subsequent analyses, steel beams and columns are going to be simulated as finite shell/beam elements and the connection behaviour is to be represented by the simplified models. This sub-frame model will also be used for parametric studies to investigate restrained beam behaviour under fire attacks with particular emphasis on the role of the connections.

ACKNOWLEDGEMENT

The research work described in this paper is part of a project funded under Grant EP/C 510984/1 by the Engineering and Physical Sciences Research Council of the United Kingdom. This support is gratefully acknowledged by the authors.

REFERENCES

[1] Y. C. Wang, *Steel and Composite Structures: Behaviour and Design for Fire Safety*, Spon Press: London and New York 2002.

[2] D. J. O'Callaghan, and M. A. O'Connor, "Comparison of finite element models of composite steel framed buildings behaviour in fire," In: *Proceedings of the First International Workshop on Structures in Fire*, Copenhagen: Denmark, 2000.

[3] M. A. O'Connor, and D. M. Martin, "Behaviour of a multi-storey steel framed building subjected to fire attack," *Journal of Constructional Steel Research*, vol. 46, no. 1/3, p. 295, 1998.

[4] M. Gillie, *Development of Generalized Stress Strain Relationships for the Concrete Slab in Shell Models*, PIT Project research Report SS1, The University of Edinburgh, 1999.

[5] M. Gillie, A. Usmani M. Rotter and M. O'Connor, Modelling of heated composite floor slabs with reference to the Cardington experiments, *Fire Safety Journal*, vol. 36, No.8, pp. 745-767, 2001.

[6] M. Sarraj, *The Behaviour of Steel Fin Plate Connections in Fire*, PhD thesis, Department of Civil and Structural Engineering, University of Sheffield, UK 2007.

[7] N. Krishnamurthy, and D. E. Graddy, "Correlation between 2- and 3- dimensional finite element analysis of steel bolted end-plate connections", *Computers and Structures*, vol. 6, pp. 381-389, 1976.

[8] A. R. Kukreti, T. M. Murray, and A. Abolmaali, "End plate connection moment-rotation relationship", *Journal of Constructional Steel Research*, vol. 8, pp. 137-157, 1987.

[9] G. J. van der Vegte, Y. Makino, and T. Sakimoto, "Numerical research on single-bolted connections using implicit and explicit solution techniques", Memoirs of the Faculty of Engineering, Kumamoto University, Japan, 2002 vol. 47, no.1, pp. 19-44.

[10] A. N. Sherbourne, and M. R. Bahaari, "3D simulation of end-plate bolted connections", *Journal of Structural Engineering*, vol. 120, no. 11, pp. 3122-3136, 1994.

[11] A. N. Sherbourne, and M. R. Bahaari, "Finite element prediction of end-plate bolted connections behaviour. I: Parametric Study", *Journal of Structural Engineering*, vol. 123, no. 2, pp. 157-164, 1997.

[12] O. S. Bursi, and J. P. Jaspart, "Benchmarks for finite element modelling of bolted steel connections", *Journal of Constructional Steel Research*, vol. 43, no. 1-3, pp. 17-42, 1997.

[13] G. J. van der Vegte, "Numerical simulations of bolted connections: the implicit versus the explicit approach", Available at http://www.bouwenmetstaal.nl/ congres_eccs_04/Vegte_Makino_bolted.pdf, 2008.

[14] ABAQUS, Standard user's manual. Hibbitt, Karlsson and Sorensen, Inc, 2006.

[15] N. Gebbeken, H. Rothert, and B. Binder, "On the numerical analysis of end plate connections", *Journal of Constructional Steel Research*, vol. 30, pp. 177-196, 1994.

[16] O. S. Bursi, and J. P. Jaspart, "Basic issues in the finite element simulation of extended end plate connections", *Computers and Structures*, vol. 69, pp. 361-382, 1998.

[17] D.P. Flanagan, and T. A. Belytschko, "A uniform strain hexahedron and quadrilateral with orthogonal hourglass control", *International Journal for Numerical Methods in Engineering*, vol. 17, no.5, pp. 679-706, 1981.

[18] R. Charlier, and A. M. Habraken, "Numerical metallisation of contact phenomena by finite element method", *Computers and Geotechnics*, vol. 9, pp. 59-72, 1990.

[19] K. J. Bathe, *Finite Element Procedures*, Prentice Hall: Englewood Cliffs, New Jersey, 1982.

[20] A. M. Kanvinde, I. R. Gomez, M. Roberts, B. V. Fell, and G.Y. Grondin, "Strength and ductility of fillet welds with transverse root

notch," *Journal of Constructional Steel Research*, vol.65, pp. 948-958, 2009.

[21] Y. Hu, J. B. Davison, I. W. Burgess, and R. J. Plank "Experimental study on flexible end plate connections in fire," In: *Proceedings of 5th European Conference on Steel Structures*, Graz: Austria, 2008; pp. 1007-1012.

[22] A. Cornec, I. Scheider, and K.H. Schwalbe, "On the practical application of the cohesive model", *Engineering Fracture Mechanics*, vol. 70, pp.1963-1987, 2003.

[23] I. Scheider, M. Schődel, W. Brocks, and W. Schőnfeld, "Crack propagation analyses with CTOA and cohesive model: Comparison and experimental validation", *Engineering Fracture Mechanics*, vol. 73, pp. 252-263, 2006.

[24] H. X. Yu, I. W. Burgess, J. B. Davison, and R. J. Plank, "Experimental Investigation of the Behaviour of Fin Plate Connections in Fire," *Journal of Constructional Steel Research*, vol. 65, pp.723-736, 2009a.

[25] H. X. Yu, I. W. Burgess, J. B. Davison, and R. J. Plank, " Tying capacity of web cleat connections in fire – part 1: test and finite element simulation," *Engineering Structures*, vol. 31, no.3, pp. 723-736, 2009b.

[26] H. X. Yu, I. W. Burgess, J. B. Davison, and R. J. Plank, "Experimental Investigation of the Behaviour of Flush Endplate Connections in Fire," *J Construct Steel Res*., vol. 65, pp. 723-736, 2009.

[27] CEN, *BS EN 1993-1-8:2005 Eurocode 3: Design of steel structures - Part 1.8: Design of joints*, European Committee for Standardisation, Brussels, 2005.

[28] Y. Hu, J. B. Davison, I.W. Burgess, and R. J. Plank, "Component modelling of flexible end-plate connections in fire," *International Journal of Steel Structures*, vol.9, pp.1-15, 2009.

[29] J. P. Jaspart, *Semi-Rigidity in Connections of Structural Steelworks: Theory, Analysis and Design*, Lecture notes for Advance Professional Training, Italy, 1999.

[30] K. Weynand, J. P. Jaspart, and M. Steenhuis "The stiffness model of revised annex J of eurocode 3," In: *Proceedings of the Third International Workshop: Connections in Steel Structures: Behaviour, Strength and Design*, Trento: Italy, 1995; pp. 441-452.

Received: December 07, 2009 Revised: January 20, 2010 Accepted: January 21, 2010