MODELLING OF FLEXIBLE END PLATE CONNECTIONS IN FIRE USING COHESIVE ELEMENTS

Ying Hu[1], Ian Burgess[2], Buick Davison[3], Roger Plank[4]

ABSTRACT

In the UK, simple steel connections, including flexible end plates, fin plates and web cleats, are the most popular for steel structures. Experimental tests completed in Sheffield have shown that issues concerning tying resistance and ductility are problematic for simple steel connections at elevated temperatures, which could significantly affect the overall performance of steel structures due to a loss of structural integrity in a fire situation.

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 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.

1. INTRODUCTION

Simple steel connections including flexible end plates (header plates), fin plates and web cleats (double web angles), are the most popular steel connection types currently in use

[1] PhD research student, Department of Civil and Structural Engineering, The University of Sheffield, Sheffield S1 3JD, UK, Email: cip05yh@sheffield.ac.uk

[2] Professor, Department of Civil and Structural Engineering, The University of Sheffield, Sheffield S1 3JD, UK, Email: ian.burgess@sheffield.ac.uk

[3] Associate Professor, Department of Civil and Structural Engineering, The University of Sheffield, Sheffield S1 3JD, UK, Email: j.davison@sheffield.ac.uk

[4] Professor, School of Architecture, The University of Sheffield, Sheffield S10 2TN, UK Email: r.j.plank@sheffield.ac.uk

for building construction in the UK, owing to the simplicity and economy in both fabrication and assembly[1]. The fire research group at the University of Sheffield developed an extensive series of tests to investigate the robustness of steel connections in a fire situation; the experimental tests demonstrated that issues concerning the resistance and ductility of connections are problematic in fire conditions. Conducting experimental tests is always time-consuming, expensive, and poses the additional difficulties of recording movement and strain within a furnace. Thereafter using experimental data for validation, but simulating the connection behaviour with finite element modelling, provides an opportunity for wider parametric investigations and eliminates the limitations associated with experiments [2]. Initial attempts to simulate steel connections started with two dimensional models (2D 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.* [3] and Kukreti *et al.* [4] 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.* [5] 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 [6] [7] 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 the hole boundary was neglected. Bursi and Jaspart [8] presented a more realistic finite element model for T-stub connections. This numerical model is capable of simulating the complex interactions such as contact, friction, stick and slip conditions, stress concentrations and prying actions in a real connection. Bolts and endplates in the simulation are represented as individual components using brick elements, and are no longer connected through common nodes, enabling relative movement between these components [9]. Although this numerical approach results in finite element simulations which are much more complicated and computationally expensive in terms of time, it has nevertheless been adopted by many researchers owing to the improvement in numerical accuracy.

A three dimensional numerical model was created for a flexible end plate connection, using the ABAQUS finite element code, in order to investigate its resistance and ductility at ambient and elevated temperatures. This model started with the creation of individual components such as bolts, endplates, beams and columns, and then assembled these components, as shown in *Fig. 1*.

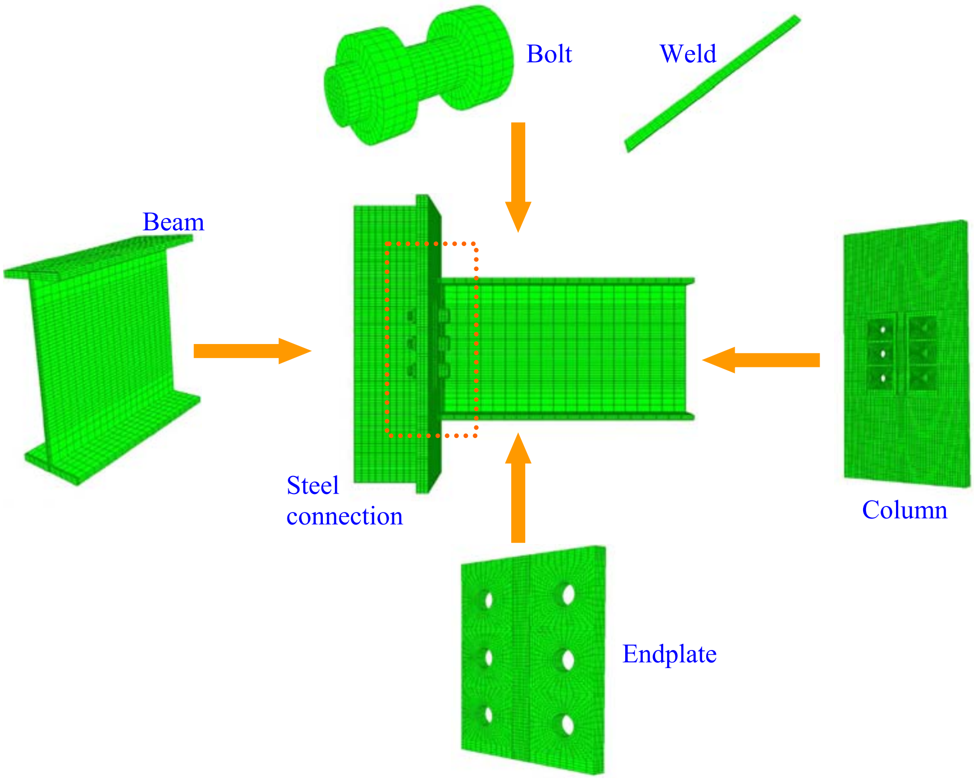


Fig. 1 - FE model for a flexible end plate connection

All these components were modelled using eight-node continuum hexahedral brick elements, and a small number of cohesive elements were used in the heat affected zone (HAZ) where the failure of endplates was seen to occur. The brick element has the capability of representing large deformation, and geometric and material nonlinearity, whilst the traction separation law of cohesive elements is able to demonstrate the rupture of end pates in a real connection. The contacts among bolts, endplates and column flanges were simulated by surface-to-surface formulations. In order to simulate this nonlinear performance, an intensive mapped mesh was made within the bolts and the vicinity of the bolt holes, shown in *Fig. 1*. The following discusses the details of how to create a FE model for a flexible end plate connection.

2.1 Solution Strategy

Within ABAQUS, two different solution strategies are available: the standard analysis and the explicit solution procedure. The standard analysis is implicitly based on static equilibrium, characterized by the assembly of a global stiffness matrix and simultaneous solution of a set of linear or nonlinear equations [9], which enables a wide range of linear and nonlinear engineering simulations to be carried out efficiently. For most nonlinear analyses, the Newton-Raphson method is used to converge the solution at each time step along the force deflection curve. However, if the tangent stiffness is zero, Newton-Raphson method is unable to achieve the convergence. To avoid this problem, the Arc-Length algorithm (Riks method) should be used to allow the load and displacement to vary throughout the time step [10]. Nevertheless, for a numerical model with complicated contact interactions, these two solution algorithms are unlikely to produce an easy and smooth solution in the computation.

The explicit solution procedure is a dynamic procedure originally developed to simulate high-speed impact events in which inertia plays a dominant role in the solution, and achieving the convergence is not needed in the simulation. This approach has proved to be valuable in solving static problems as well. One advantage of the explicit procedure over the implicit procedure is the greater ease in resolving complicated contact problems. In addition, for very large models, the explicit procedure requires less system resources than the implicit procedure [10].

Applying the explicit dynamic procedure for quasi-static simulations requires some special considerations. Since a quasi-static event is a long-time process, it is often computationally impractical to have the simulation in its natural time scale, which would require an excessive number of small time increments[10]. Hence, this event must be accelerated in some way in the simulation; however, the arising problem is that inertial forces (dynamic effects) become more dominant as the event is accelerated. So the crucial point for a quasi-static simulation is to model the event in the shortest time period in which inertial forces remain insignificant. To achieve this point, Vegte [9] recommends researchers to monitor the various components of the energy balance throughout the loading process. In a quasi-static simulation, the work applied by the external forces is nearly equal to the internal energy of the system; as a general rule, the kinetic energy of the deforming material should not exceed a small fraction (typically 5% to 10%) of its internal energy throughout most of the process [10]. In order to reduce the solution time in simulations, *mass scaling* (artificially increase the mass to reduce inertial effects) is the only option for researchers, which enables an analysis to be performed economically without artificially increasing the loading rate [10].

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.* [4] and Gebbeken *et al.* [11] 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 is too stiff for the representation of the real deformations [11], and the hexahedron (brick) element is 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 [13] 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 [13] 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 [10]. (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 [10], which can often make the elements unusable unless it is controlled. In order to control the hourglass modes in elements, Flanagan and Belytschko [14] 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 [10]. 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 [8]. 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 [8] 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 for a finite element model.

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 [8]. 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 [9].

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 to steel elements [15]. However, it is of great importance to be careful with the assignment of the slave and master surfaces [2]. It is generally accepted that the surfaces working as master surfaces should belong to the bodies with the stronger material or a finer mesh. In simulating bolted steel connections, experience shows that heat generation caused by frictional sliding is not significant in experimental tests and therefore may be ignored in the finite element modelling.

2.4 Material Properties for the Finite Element Model

For realistic simulations, Bursi and Jaspart [13] state that proper material properties are required in the explicit solution procedure. The material properties for the various components of steel connections may be determined from the engineering stress-strain relationship using nonlinear material curves recommended in Eurocode 3. They may also be defined according to stress-strain relationships obtained in standard tensile tests of steel.

In the connection tests, a 254UC89 was used for the column and a 305x165UB40 for the beam. The thickness of the end plate was 10 mm. The steel used was S275 for endplates and UB sections; and S355 was used for UC sections. All the bolts are M20 grade 8.8 used in 2 mm clearance holes. The nominal material properties of these components are summarised in *Table 1*.

Table 1 - Material properties

Material type	Yield stress [N/mm^2]	Ultimate stress [N/mm^2]	Density [kg / m^3]	Young's modulus [kN/mm^2]	Poisson's ratio
S275	275	450	7850	205	0.3
S355	355	550	7850	205	0.3
8.8 bolt	640	800	7850	205	0.3

However, material properties used for FE modelling are between the tensile test data from testing labs and the material properties determined according to Eurocode 3, which is shown as a green curve in Fig. 2 (blue and red curves in this figure respectively represent the material properties determined from tensile tests and Eurocode 3).

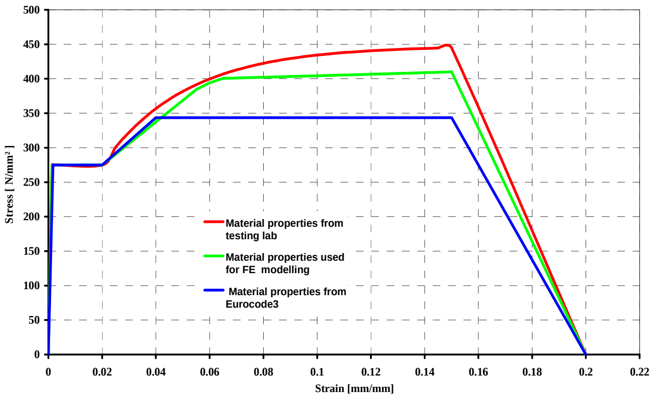


Fig. 2 - Material properties for steel

Since ABAQUS codes operate in a large deformation setting, in order to consider the deformed area the nonlinear relationship of *true stress* versus *true strain* is required to be defined for steel components. However, most material test data are supplied with engineering stresses and strains (nominal stresses and nominal strains) according to the uniaxial material testing response [16]. In such situations, it is necessary to convert material data from engineering stress and strain to true stress and strain using the following relationship:

$$\sigma_{true} = \sigma_{nom} (1 + \varepsilon_{nom}) \quad (1)$$

σ_{true} is the true stress
σ_{nom} is the nominal stress
ε_{nom} is the nominal strain

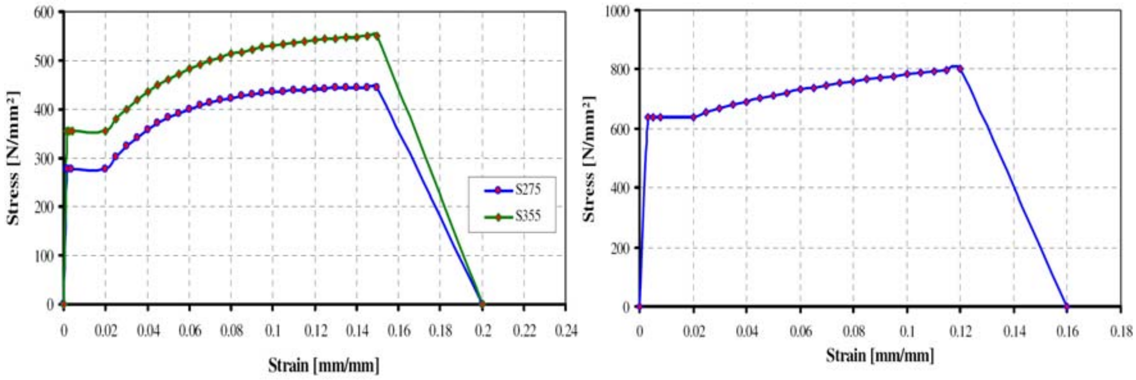
The relationship between the true strain and nominal strain is defined as:

$$\varepsilon_{true} = \ln (1 + \varepsilon_{nom}) \quad (2)$$

The true stress (σ_{true}) is a function of the nominal stress and nominal strain; and the true strain (true total strain, ε_{true}) is determined by the logarithm of nominal strain (total strain, ε_{nom}). For inputting into ABAQUS, the total strain values (ε_{true}) should be decomposed into the elastic and plastic strain components ($\varepsilon_{el, true}$ and $\varepsilon_{pl, true}$). The true elastic strain ($\varepsilon_{el, true}$) can be captured by the true stress (σ_{true}) divided by the Young's modulus (E); and the true plastic strain, required for the explicit solution procedure, can be obtained using the following relationship:

$$\varepsilon_{pl,\,true} = \varepsilon_{true} - \varepsilon_{el,\,true} = \ln(1 + \varepsilon_{nom}) - \sigma_{true}/E \qquad (3)$$

Hence, the elastic-plastic material curves, shown in Figures 2 and 3, are used for the aforementioned steel components in the connection simulations.



(a) (b)

Fig. 3 - Stress-strain curves for (a) S275 and S355 steel and (b) grade 8.8 high strength bolts [2]

2.5 Modelling the Rupture with Cohesive Elements

To simulate the rupture of endplates, a small number of cohesive elements have been embedded into the heat affected zone (HAZ) in the numerical model. When the cohesive elements and their neighbouring components have matched meshes, it is straightforward to connect cohesive elements to other elements in a model simply by sharing nodes. If the neighbouring elements do not have matched meshes, ABAQUS enables the cohesive elements to be connected to other components by using surface-based tie constraints [10].

The cohesive elements (cohesive zone) represent a fracture of a material as separation across surface and the constitutive response of these elements is determined by the relationship of traction versus separation (*traction separation law*). The available traction-separation model in ABAQUS assumes initially linear elastic behaviour followed by the initiation and evolution of damage. To determine this constitutive response, a number of parameters, such as critical separation (δ_0), cohesive energy (Γ_o) and cohesive strength (T_0), are required for the explicit solution procedure, as shown in *Fig. 4* .

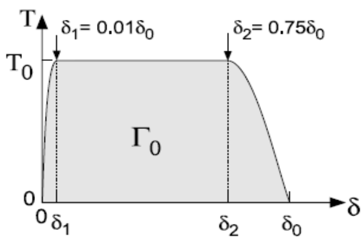


Fig. 4 - Traction-separation law for fracture [12]

Cornec *et al.* [12] recommend that the cohesive strength (T_0) may be taken as the maximum stress at fracture in a round notched tensile bar. But Scheider *et al.* [17] add that this procedure might not be applicable to thin specimens, as round notched bars can not be machined from sheet metal and the individual failure mode (normal fracture) would be different from that in the flat specimen (slant fracture). As an estimation for the simulation, Scheider *et al.* [17] recommend that the nominal stress of the flat tensile specimen at fracture (load divided by the area of the normal projection of its inclined fracture surface, $F_{frac}/A_{frac} \approx 470.5$ Mpa) may be used as T_0 . To simulate the progressive damage in the cohesive zone, it is of great importance to determine the critical separation (δ_0) and cohesive energy (Γ_0). The determination of δ_0

heavily relies on experience in numerical simulation and experimental tests, and three times the separation value at damage initiation (δ_1, shown in *Fig.*4) or over has been adopted for the explicit solution procedure. Thus the cohesive energy (Γ_0) may be estimated by using the following relationship [12]:

$$\Gamma_o = 0.87\, T_0\, \delta_0 \quad (4)$$
or
$$\Gamma_o = T_0\, \delta_0 \quad (5)$$

3. COMPARISON OF NUMERICAL RESULTS WITH EXPERIMENTAL TESTS

The numerical simulations were validated against experimental results at ambient temperatures and also at elevated temperatures. Experimental data was taken from connection test results carried out at the University of Sheffield [18].

3.1 Comparison of Flexible End Plate Model at Ambient Temperatures

Hu *et al.* [18] investigated the resistance and rotation capacity (ductility) of simple steel connections at ambient and elevated temperatures, focusing on flexible end plates. In the programme, twelve tests have been performed for end plate connections, including three tests at ambient temperatures and nine tests for high temperatures. The deformation (rotation) in the connection zone was recorded by inclinometers (angular transducers) for the first three tests at ambient temperatures, and the applied external force was captured by strain gauges on the loading system (three Macalloy bars: oven, link and jack).

The numerical model was created for flexible end plates by using the ABAQUS commercial software package, and the geometrical details of the model are shown in *Fig. 5*. A 254UC89 was used for the column and a 305x165UB40 for the beam, and the thickness of endplate is 10 mm. The steel used was S275 for universal beams and end plates, whereas the column was S355. Dimensions in *Fig. 5* are shown in mm.

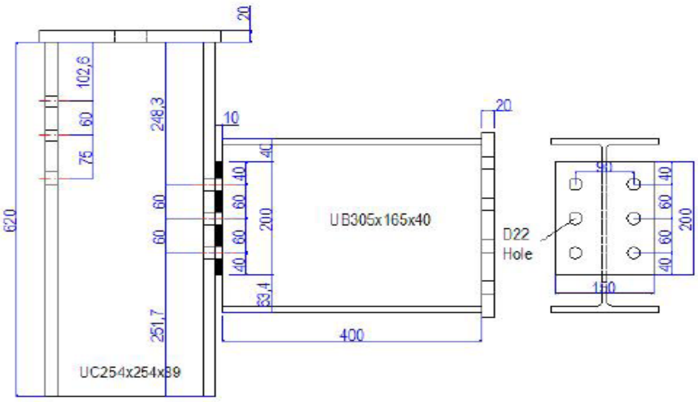


Fig. 5 - Geometrical details of the flexible endplate connection

The deformed and un-deformed shapes of the numerical model are displayed for flexible end plates in *Fig.6*, including the contour plots for the components as well, such as bolts and endplates. The FE analysis clearly demonstrates that the rotation capacity of these connections is mainly produced by deformation in the end plates, welds and bolts, and the deformation of the column flange and beam web may be neglected in the analysis.

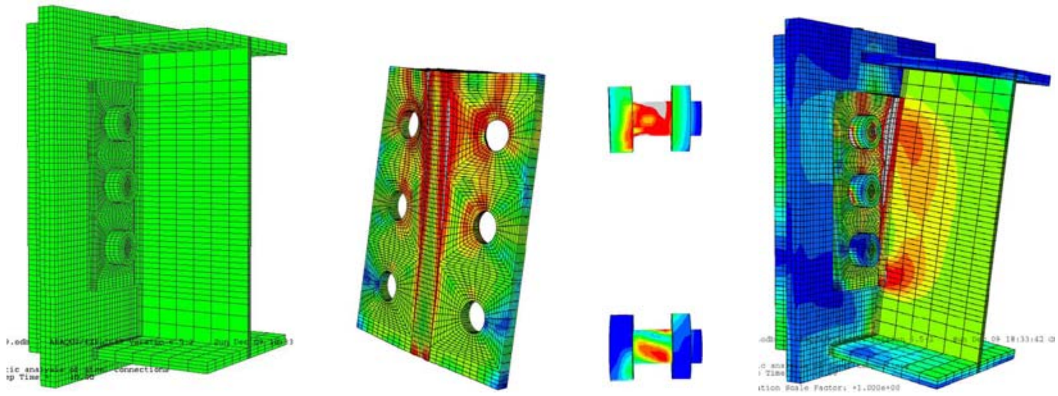
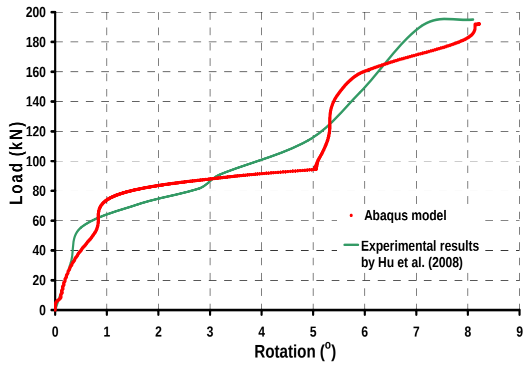
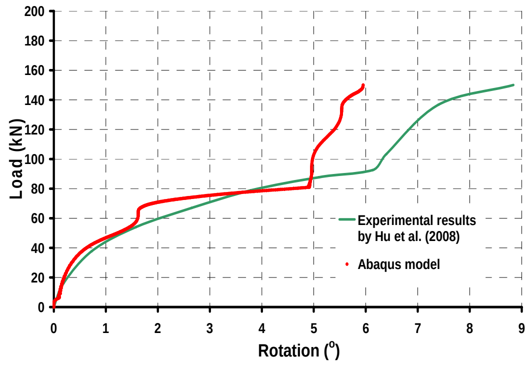


Fig. 6 - FE model of flexible end plate connection: deformed and un-deformed shapes

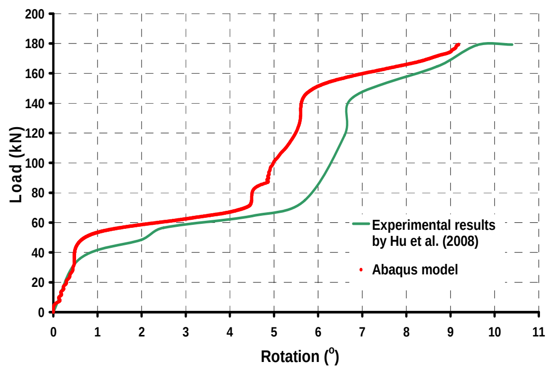
The relationships of loads and rotations of the numerical model have been compared with experimental data of three connection tests, as shown in *Fig. 7* . The red curves in *Fig. 7* (a), (b) and (c) are the numerical plots produced in ABAQUS; and the loads and rotations, recorded in experimental tests, are displayed as green. The kink in the green curve is at about 6^o rotation as an evidence of the beam bottom flange contacting with the column flange. It is apparent in *Fig7.* (a) and (c) that the numerical plots are in good agreement with experimental plots, and also noted in *Fig7.* (b) that the discrepancy exists between the numerical and experimental results, as variety between the real specimens cannot be represented in a numerical model.



(a) 35^o



(b) 45^o

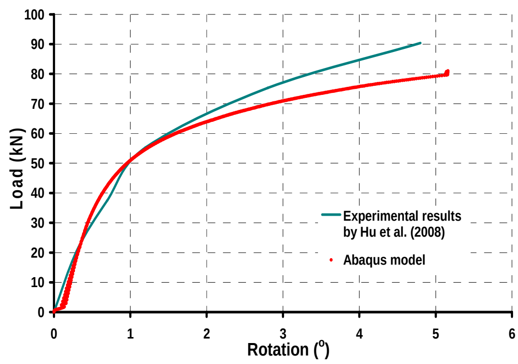


(c) 55^o

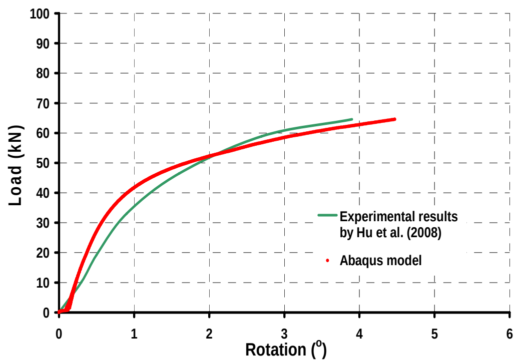
Fig. 7 - Load-Rotation comparisons between FE model and experimental results for flexible end plate connections (a) 35^o (b) 45^o (c) 55^o

3.2 Comparison of Flexible End Plate Model at Elevated Temperatures

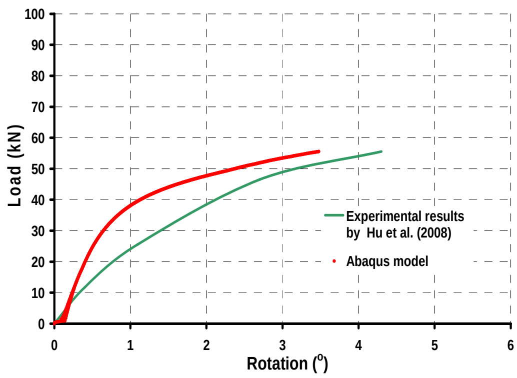
Nine tests for flexible end plates have been carried out at the temperatures of 450 °C , 550 °C and 650 °C, and the relationships of loads versus rotations are plotted in *Fig. 8*. To simulate the performance of these connections in fire conditions, the numerical model requires the material properties to be applied at the predetermined temperatures, and the reduction retention factors used are recommended from EC3 (BSI, 2005). The deformed shapes of connections are also shown in *Fig. 8* for each temperature. It was observed that the connections both in numerical simulation and experiments failed by the rupture of endplates before the beam flange contacted with the column flange. Except for *Fig. 8* (g), the curves of loads and rotations, produced by numerical simulation, are in good agreement with recorded experimental data.



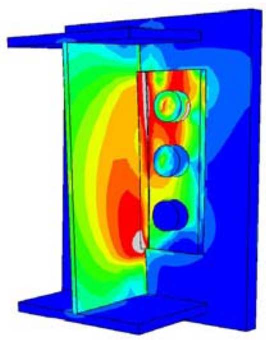
(a) 35° - 450 °C



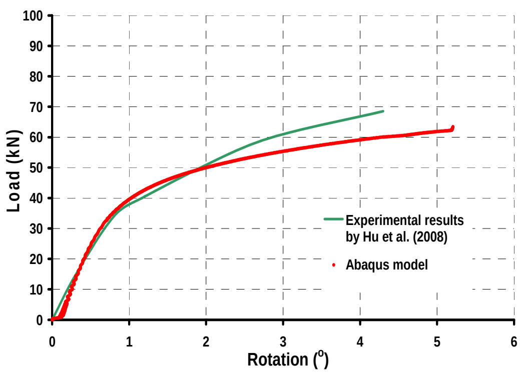
(b) 45° - 450 °C



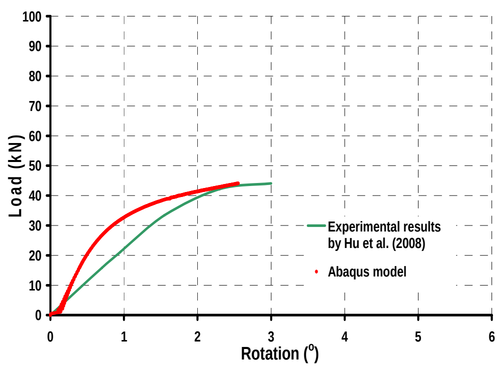
(c) 55° - 450 °C



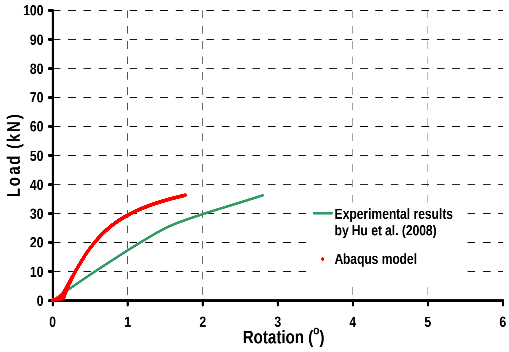
(d) Deformed shape at 450 °C



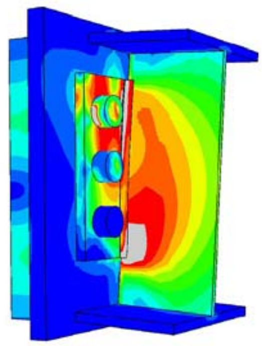
(e) 35° - 550 °C



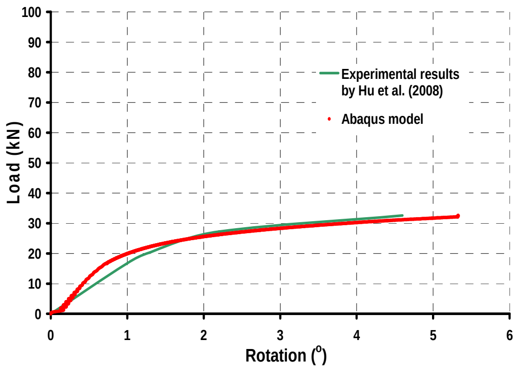
(f) 45° - 550 °C



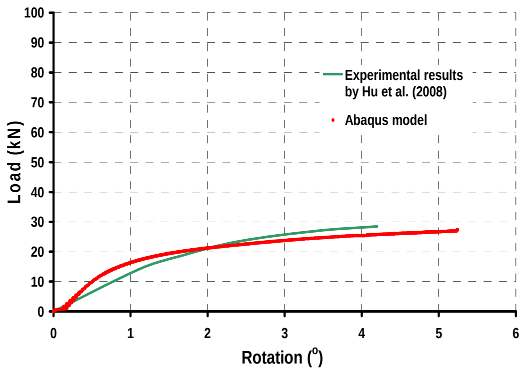
(g) 55° - 550 °C



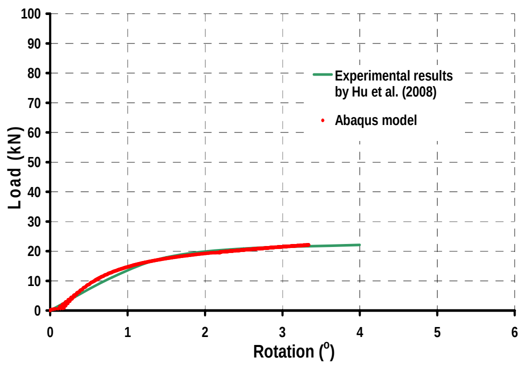
(h) Deformed shape at 550 °C



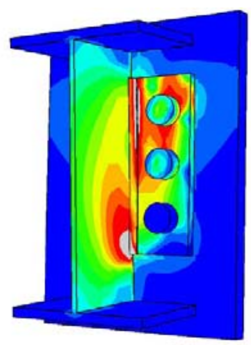
(i) 35° - 650 °C



(j) 45° - 650 °C



(k) 35° - 650 °C



(l) Deformed shape at 650 °C

Fig. 8 - Load-Rotation comparisons between FE model and experimental results for flexible end plate connections

4. CONCLUSIONS

This paper reported on the development of a finite element model embedded with cohesive elements to estimate the resistance and ductility of flexible endplate connections in fire conditions. From the comparative results in *Fig. 7* and *Fig. 8*, the numerical model with cohesive elements is able to estimate the failure of steel connections due to the rupture of endplates. In addition, the results from the aforementioned two figures also showed that the explicit solution technique is a reliable and suitable tool to effectively simulate the performance of bolted connections. Therefore, the simulation strategies employed in this paper may be use for further parametric studies of flexible endplate connections.

5. 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.

6. REFERENCES

[1] Steel Construction Institute, "Joints in Simple Construction", SCI, Berkshire, Publication No. P206, 2002

[2] Sarraj, M. " The Behaviour of Steel Fin Plate Connections in Fire ", PhD Thesis, The University of Sheffield, 2007

[3] Krishnamurthy, N. and Graddy, D.E. , "Correlation between 2- and 3- Dimensional Finite Element Analysis of Steel Bolted End-Plate Connections", *Computers and Structures*, Vol. 6, pp. 381-389, 1976

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

[5] Vegte, G.J. van der, Makino, Y. and Sakimoto, T., "Numerical Research on Single-Bolted Connections Using Implicit and Explicit Solution Techniques", Memoirs of the Faculty of Engineering, Kumamoto University, Vol. 47, No.1, 2002

[6] Sherbourne, A.N. and Bahaari, M.R. , "3D Simulation of End-Plate Bolted Connections", *Journal of Structural Engineering*, Vol. 120, No. 11, pp. 3122-3136, 1994

[7] Sherbourne, A.N. and Bahaari, M.R. , "Finite Element Prediction of End-Plate Bolted Connections Behaviour. I : Parametric Study", *Journal of Structural Engineering*, Vol. 123, No. 2, pp. 157-164, 1997

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

[9] Vegte, G.J. van der, "Numerical Simulations of Bolted Connections: The Implicit Versus The Explicit Approach", *@http://www.bouwenmetstaal.nl/congres_eccs_04/ Vegte_Makino_bolted.pdf* , 2008

[10] ABAQUS analysis user's manual version 6.6, ABAQUS inc 2006

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

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

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

[14] Flanagan, D.P. and Belytschko, T.A. , "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

[15] Charlier, R. and Habraken A.M. , "Numerical metallisation of contact phenomena by finite element method" , *Comput. Geotech.*, Vol.9, pp. 59-72, 1990

[16] Bathe, K. J. , "Finite Element Procedures", Prentice Hall, Englewood Cliffs, New Jersey, 1982

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

[18] Hu, Y., Davison, B., Burgess, I. and Plank, R. , "Experimental Study on Flexible End Plate Connections in Fire", Eurosteel 2008: 5th European Conference on Steel and Composite Structures, in press, 2008