PROTECTED STEEL AND COMPOSITE CONNECTIONS IN FIRE

Simulation of the mechanical behaviour of steel and composite connections protected by intumescent coating in fire

Peter Schaumann [a], Thomas Kirsch [b]

[a] Leibniz University Hannover, Institute for Steel Construction, Hannover, Germany
[b] Ingenieurbüro Thor-Schipper-Schween, Lohne, Germany

Abstract

Actual developments in numerical simulations of the structural behaviour in fire situation are focussed on taking into consideration the interaction of all structural members in a global approach. Therefore it is necessary to simulate the load bearing behaviour of connections.
With this motivation, the authors conducted experiments and thermal FE-simulations on two different connection types. In this paper, the accompanying mechanical FE-simulations of both investigated connection types will be described. The joints are defined as an end plate connection in a steel structure and a fin plate connection in a composite structure. Besides the validation of the numerical models, the results of the described investigations show that it is possible to activate a significant moment resistance within fin plate connections of composite structures. The main requirement for this activation is sufficient reinforcement strength.

Keywords: end-plate, fin-plate, bolted connection, fire safety, intumescent coating

INTRODUCTION

Actual developments in numerical simulations of the structural behaviour in fire situation are focussed on taking into consideration the interaction of all structural members in a global approach. Therefore it is necessary to describe the behaviour of connections in detail.
The first step to describe this connection behaviour is to investigate the connection experimentally. This has been conducted by many authors at ambient temperature, beginning in the early 20^{th} century (Wilson et al, 1917). Since then, many specialised test setups have been developed to investigate different connection details. Even experiments at elevated temperature and in fire have been conducted in a high number. For example in (Armer et al, 1994) the connection behaviour during some of the Cardington tests is described. Experiments focussed on the connection behaviour have been conducted by (Al-Jabri, 1999), (Wang et al, 2007) and (Schaumann et al, 2008) to mention but a few.
In addition, numerical investigations have been conducted for some elevated temperature tests. For example in (Sarraj et al, 2007) a numerical model of fin plate connections has been developed. In (Yu et al, 2008) a simulation of a steel connection using explicit analysis was presented. The explicit equation solver algorithm was found to be an alternative to the standard algorithm especially if large deformations occur. The authors of this paper also published an FE-analysis and showed that a general damage algorithm can help to increase the accuracy of simulations reasonably (c.f. (Schaumann et al, 2011)).
Although much knowledge has been gained from all these tests and simulations, there are still some unrealistic simplifications concerning the temperature field. In many tests the connection is heated to a constant temperature and then increasingly loaded. As the transient temperature field in a connection is not uniform in a fire situation, this is not realistic. So in other tests the connection is heated following ISO-fire curve or a natural fire curve. Although usual steel constructions are protected against fire to reach a sufficient fire resistance time, most tests are conducted with bare steel connections. For this reason the experiments that underlay this paper have been conducted with protected steel and composite connections.

1 EXPERIMENTS

As described, two full scale fire tests have been conducted. The first test dealt with an extended end plate connection within a steel structure and the second was on a fin plate connection within a composite structure. Both connections are displayed in Fig. 1.

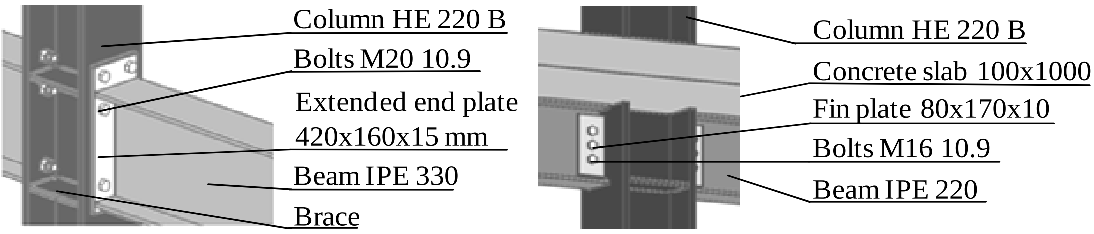

Fig. 1 Scheme of tested connections

As tests and specimen are described in detail in former publications (c.f. (Schaumann et al, 2012) and (Schaumann et al, 2012a)) they will only be mentioned shortly. All steel members consist of S235 steel. The bolts are 10.9 high strength bolts. The concrete slab consists of C25/30 concrete within two layers of Q188 meshes of S500 reinforcing steel. The slab is fully connected to the steel beams by headed studs. All materials have been tested on tensile or compressive strength at ambient temperature. The results are shown in Tab. 1.

Tab. 1 Yield stress and tensile strength of components of tested connections [MPa]

		End plate connection			Fin plate connection				
	Column	Beam	Plate	Bolts	Beam	Plate	Bolts	Concrete	Reinforcement
Yield stress	336	326	314	1039	385	336	1031	39 (f_{ck})	600
Tensile strength	465	443	447	1154	506	469	1145	3 (f_{ct})	639

As can be seen in Fig. 2 for the end plate connection, the specimen are located inside a load structure with massive steel plates on top. The load of the connection is constant. It is M=38.1 kNm (μ=0,5) for the end plate and M=20.5 kNm (μ=0,7) for the fin plate connection.

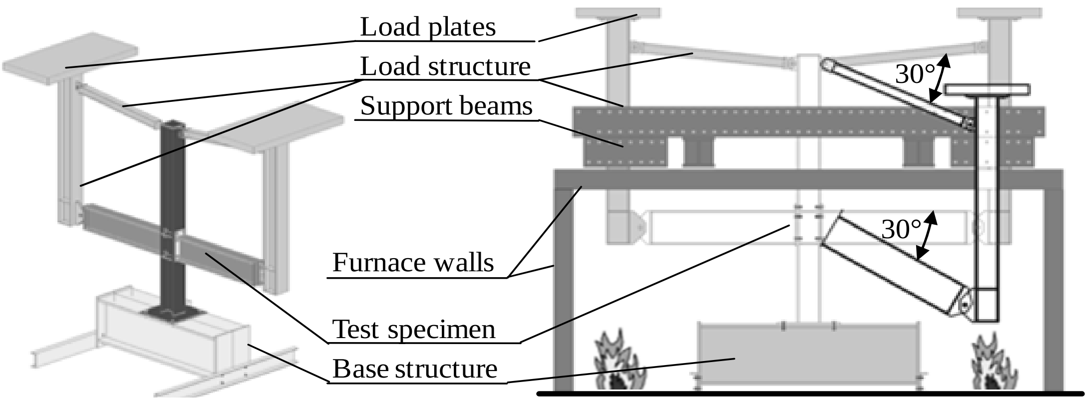

Fig. 2 Scheme of test setup

After the constant loads are applied to the specimen, the furnace temperatures are increased following the ISO-fire curve until connection failure. To ensure a realistic temperature field, the connection and surrounding parts of the specimen are protected by intumescent coating for the fire resistance class R 30. The measured average thickness of the painting is 1 mm.

During the tests, gas and specimen temperatures are measured by 54 thermocouples. The beam deflection is measured by potentiometers. Thickness of the intumescent coating is measured using a furnace camera and small steel bars used as visual gauges.

The experimental results are described in detail in (Schaumann et al, 2012) and (Schaumann et al, 2012a). In this paper, the results are shown within the validation of the FE-simulations.

2 GEOMETRY AND BOUNDARY CONDITIONS OF FE-ANALYSIS

To simulate the mechanical connection behaviour in fire, a detailed 3D numerical model of each connection has been developed with the software Abaqus. The geometry of the models is build by solid brick elements (C3D8R) and generally equals the test specimen. Following, only differences between model and specimen are described. One of these differences is the use of the specimen symmetry to reduce the model size. This can be seen in Fig. 3 for the example of the fin plate connection. Another simplification is the use of 1D beam elements for the steel/composite beam (B31) from 400 mm distance from the connection (c.f. Fig. 3). The same elements are used for the reinforcement.

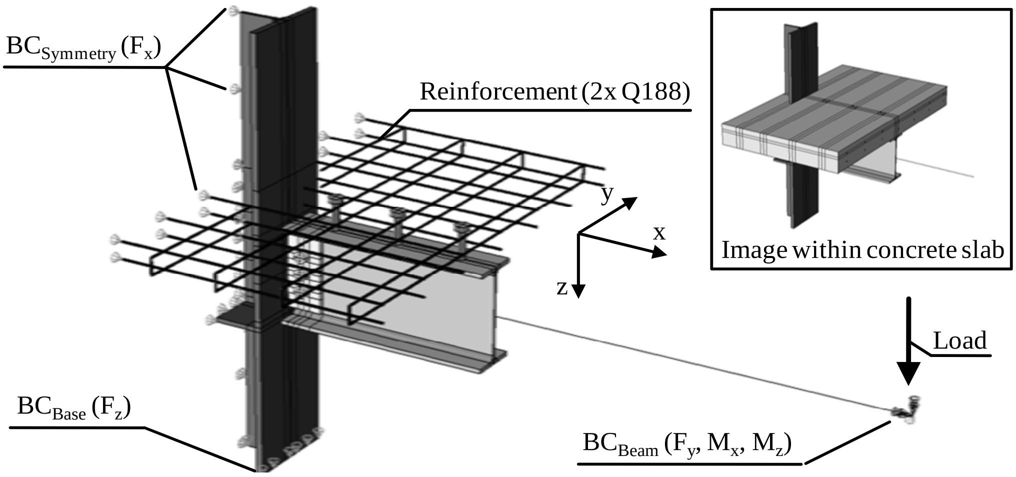

Fig. 3 Numerical model of fin plate connection within boundary conditions (BC)

To simulate the boundary conditions, the model is pinned in x-direction in the whole symmetry area and in z-direction at the bottom of the simulated part of the column (c.f. Fig 3). The end of the lever arm is also pinned in y-direction and the rotation around x- and z-direction is blocked in accordance with the connection between specimen and load structure.
The interactions inside the model are simulated using “constraints” between welded members and between the two beam parts. Most other adjacent members (e.g. bolts and fin plate) are connected by a general interaction with a friction coefficient of μ=0.3 and the ability to separate from each other. The reinforcement is defined as “embedded” in the concrete slab.
As described before, the explicit analysis is preferable to simulate connections with large deformations. So this kind of analysis is used. To ensure a stable time increment of $t=5\cdot10^{-7}$, the element edge size is chosen to be larger than 5 mm. As the model volume of the connection within the concrete slab is higher, element sizes are doubled in parts (c.f. Fig. 3).
During the tests it has been observed that failure of one end plate connection occurred by buckling of the lower beam flange. As buckling does not occur in a perfect geometry, the model geometry is set to be imperfect. This is realised using an Eigenvalue-analysis. After this analysis, the local buckling mode with the lowest Eigenvalue is adopted with a maximum deflection of u_{max}=1 mm. The Buckling mode (with u_{max}=10 mm) is shown in Fig. 4.

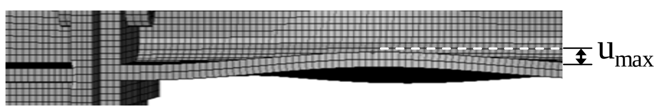

Fig. 4 Numerical model of fin plate connection

As the experiments are conducted in two steps, the numerical simulation follows the same procedure. First the loads are applied using the smooth-step algorithm of Abaqus. Second the temperature is increased. As deflections have only little influence in the temperature field, the thermal simulation is conducted separately. So the temperature field is calculated in advance, taking into account the nonlinear behaviour of the intumescent coating (c.f. (Schaumann et al, 2012) / (Schaumann et al, 2012a)) and is applied to the mechanical simulation afterwards.

3 MATERIAL PROPERTIES OF FE-ANALYSIS

The properties of the used materials are investigated experimentally at ambient temperatures (see Tab. 1). To extend the results for elevated temperatures, Eurocode regulations are used.
So for Steel (including reinforcement and bolts), regulations in (Eurocode 3-1-2, 2010), 3.2, based on (Schaumann et al, 1984), are used to determine a stress-strain-behaviour from yield stress at ambient temperature. Differing from the Eurocode, the ultimate stress is added at a strain of ε=0.15 and connected linearly to the stress at ε=0.02 for a better correlation with test results (c.f. Fig. 5). The reduction of the ultimate stress over temperature is realised using results in (Renner, 2005). The resulting stress-strain-diagram for temperatures up to 1000°C is displayed in Fig. 5. While in this figure the nominal stress and strain are displayed for a better comparability with the test results, in the simulation the real stress and strain values are used.

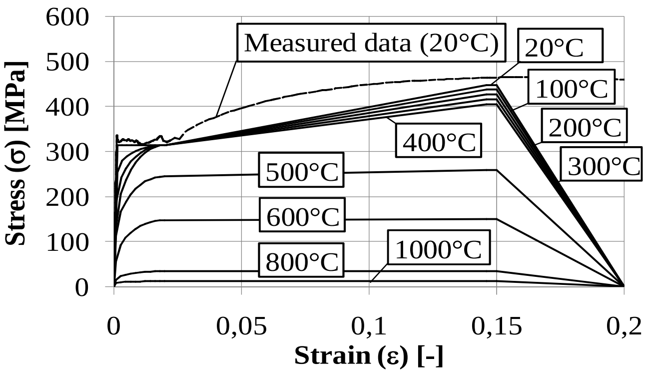

Fig. 5 Temperature dependent stress-strain relationship of end plate

The decreasing parts of the stress values in the diagram are not included in the material behaviour. Instead, the fracture algorithm, presented in (Schaumann et al., 2011) is used.
In the simulation of the fin pate connection, a concrete slab is included and so the material concrete has to be determined as well. The uniaxial compressive behaviour is determined using compressive strength tests, extended for elevated temperatures using regulations in (Eurocode 2-1-2, 2010). For the uniaxial tensile behaviour an approximation in (Hothan, 2004) with a linear increase and bilinear decrease of stress over strain is used.
Further input values for the “concrete damaged plasticity” material law are set to standard values with two deviations. First is the value for dilation angle, defined as ψ=30° (c.f. (Hothan, 2004)). Second is the relationship between uniaxial and biaxial concrete behaviour, which is defined by the following equation, based on test results in (Ehm, 1986).

$$(f_{ck,bi} / f_{ck}) = 2{,}27 \cdot 10^{-6} \cdot \theta^2 - 7{,}31 \cdot 10^{-4} \cdot \theta + 1{,}16 \quad (1)$$

where θ is the temperature [°C]

4 VALIDATION OF FE-ANALYSIS

The numerical models are tested and validated against the experimental results. Following the comparison between numerical and experimental results are described for each connection.

4.1 End Plate Connection

During the end plate connection test, the specimen behaved as follows. During the first 10 min, the intumescent coating inflated (c.f. (Schaumann, 2012)), leading to a less fast increase of temperatures afterwards (c.f. Fig 6). With increasing temperatures, the global deflection started to increase as well. During the first 60 min the deflection kept rather low (less than 10 mm). After 60 min of fire exposure the deflection of both beam ends increased exponentially until the first connection failed after 71 min. This is shown in Fig. 6. As can be seen in the diagram as well, the calculated global deflection of the numerical model is similar.

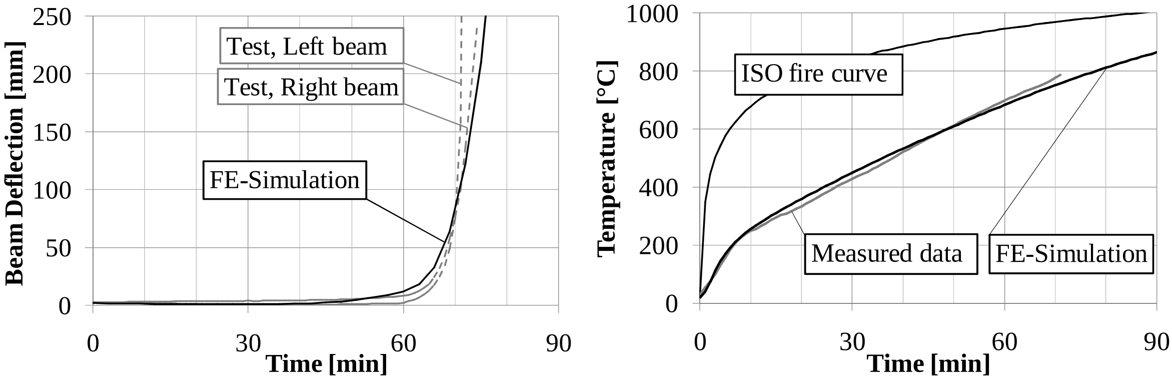

Fig. 6 Global deflection and lower flange temperature of end plate connection during fire

The reason for the connection failure in the experiment and in the FE-analysis was buckling of the component lower flange. As the furnace was stopped for safety reasons with failure of that first connection, the second connection did not fail completely. Nevertheless, the connection kept on deflecting after the end of the test and showed an upcoming combined failure of the lower flange in buckling and the end plate in bending. This occurring failure can be seen in Fig. 7 (right), where the connection after the test is depicted. It can be seen as well that the failure mode of the numerical model is similar. The lower flange is starting to buckle in the same way as seen in the experiment. The loss of contact between end plate and column is less distinct in the simulation but still there.
Moreover, within parametric studies it was found that a slight reduction of the end plate thickness changed the failure mode to end plate bending within bolt fracture. The same failure mode occurred when increasing the load at ambient temperature.
The main result of the investigation concerning the end plate connection is the shown possibility to describe the nonlinear behaviour of the connection using the finite element simulation. Moreover it is shown, that the failure mode of the connection can change due to fire situation and due to change of connection geometry.

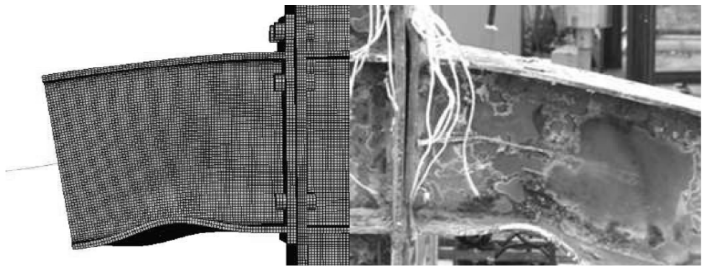

Fig. 7 FE-model just before failure and test specimen after experiment

4.2 Fin Plate Connection

In contrast to the investigated end plate, the fin plate connection is usually approximated as a pinned connection at ambient temperature. As a large rotation - exceeding service ability demands - is needed to activate the connections moment resistance, this is reasonable. So it was one aim of the test to show the possibility to activate this moment resistance in fire.
One of the fin plate connections already failed during the loading phase. The unexpected reason for this was found to be the missing ductility of the continuous reinforcement that failed before the lower beam flange got in contact with the column. For this reason the remaining connection was loaded with a reduced moment of M=20.5 kNm.
As can be seen in Fig. 8, the remaining connection resisted the ISO fire for 85 min. Up to 60 min the deflection was neglectible. Afterwards it increased exponentially, until failure occurred due to failure of the pre-damaged reinforcement.

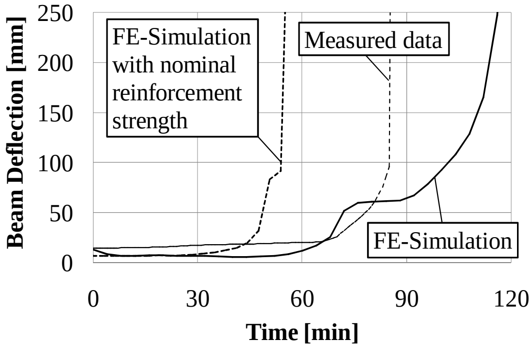

Fig. 8 Beam end deflection in fire test and FE-simulations

As can be seen in Fig. 8 as well, the finite element model behaves slightly different. In the beginning, the deflection decreases with increasing temperatures because of an elongation of the steel beam below the cooler concrete slab. This decrease of deflection was not observed in the experiment for the reason of the pre-damage. After 60 min the FE-model starts to deflect increasingly, as the specimen did in the test.
After about 75 min after ignition, the deflection stops in the FE-model. The reason for this is the contact between lower beam flange and column. Due to this contact, the moment resistance increases significantly. This behaviour was not observed in the experiment, as the contact between column and beam existed from the beginning of the test for the reason of the pre-damage of the reinforcement. Moreover the failure of the pre-damaged reinforcement was the reason of the connection failure in the experiment. So the remaining result of the test is: Even with pre-damaged reinforcement, the connection withstands the fire situation for a significant period. The FE-simulation matches with this result and shows an achievable fire resistance time of nearly 120 min. The final failure of the connection in the FE-simulation is buckling of the lower beam flange (c.f. Fig. 9).
Based on the FE-calculation, it can also be shown that reinforcement resistance is critical for this increase of the moment resistance. When reducing the yield- and ultimate stress of the reinforcement to nominal values, the fire resistance time is reduced to 70 min (cf. Fig 8).

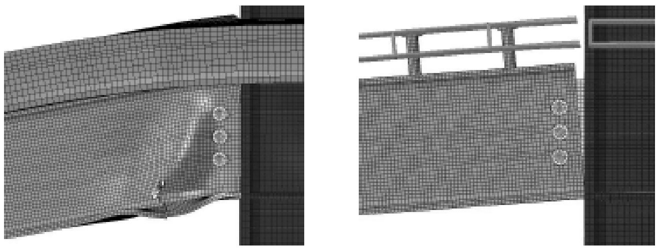

Fig. 9 Failure in FE-model with original (left) and reduced (middle) reinforcement strength

The reason for this can be seen in Fig. 9. As the reinforcement fails before contact between lower beam flange and column exists, the increase of the moment resistance does not occur. The same failure mechanism was found during the loading phase before the fire test.

5 SUMMARY AND ACKNOWLEDGEMENT

In this paper experimental and numerical investigations in the load-rotation behaviour of two protected connections in fire situation are presented. The first test specimen is an end plate connection in a steel structure. The second is a fin plate connection in a composite structure.
Concerning the first connection it was found that there is a good correlation between numerical investigation and experiment. For this reason, it is possible to use the FE-model to determine the moment-rotation-behaviour for variable connection geometries. Furthermore the model can be used to develop and validate component-method-models for this connection.

Concerning the fin plate connection an unexpected phenomenon has been identified in the load introduction phase of the test. Due to low ductility of the reinforcement, failure occurs before reaching the full plastic bending moment capacity. Using the FE-model in combination with the experiment, the authors were able to show, that this strong moment resistance can be achieved if the design of the reinforcement is improved accordingly.

The IGF-Project No. 16586 N from FOSTA is funded by the German "Federal Ministry of Economics and Technology" via AiF.

Federal Ministry of Economics and Technology

REFERENCES

Al-Jabri, K. S., The behavior of steel and composite beam-to-column connections in fire, Ph.D.-Thesis at the Department of Civil and Structural Engineering, University of Sheffield, Sheffield, 1999.

Armer, G. S. T., Moore, D. B., Full-scale testing on complete multi-storey structures. The Structural Engineer 72 (1994) 30-51, Elsevier, 1994.

Ehm, C., Versuche zur Festigkeit und Verformung von Beton unter zweiaxialer Beanspruchung und hohen Temperaturen, Dissertation, Institut für Baustoffe, Massivbau und Brandschutz, TU Braunschweig, Braunschweig, 1986.

Eurocode 2: Design of concrete structures - Part 1-2: General Rules – Structural fire design; German version EN 1992-1-2:2004 + AC:2008, Beuth Verlag, Berlin, 2010.

Eurocode 3: Design of steel structures - Part 1-2: General Rules – Structural fire design; German version EN 1993-1-2:2005 + AC:2009, Beuth Verlag, Berlin, 2010.

Hothan, S., Zur dreidimensionalen Simulation von Deckentragwerken im Brandfall, Dissertation, Institute for Steel Construction, University Hannover, Hannover, 2004.

Renner A., The effect of strain-rate on the elevated-temperature behavior of structural steel, Research dissertation, University of Sheffield, 2005.

Sarraj M., Burgess I. W., Davison J. B., Plank R. J., Finite element modeling of steel fin plate connections in fire, Fire Safety Journal 42 (2007) 408-415, Elsevier, 2007.

Schaumann, P., Zur Berechnung stählerner Bauteile und Rahmentragwerke unter Brandbeanspruchung, Dissertation, Institut für konstruktiven Ingenieurbau, Ruhr-Universität Bochum, Technisch-wissenschaftliche Mitteilungen Nr. 84-4, Bochum, 1984.

Schaumann P., Zhao B., Bahr O., Renaud, C., Fire performance of external semi-rigid composite joints, Proceedings of 6th International Conference Structures in Fire, Singapore, 2010.

Schaumann, P., Kirsch, T., Fracture simulation in a steel connection in fire, Proceedings of Conference: Applications of Structural Fire Engineering, Prague, 2011.

Schaumann, P., Kirsch, T., Timmen, F., Full scale fire test and numerical simulation of a steel connection, Proceedings 7th International Conference on Structures in Fire, Zurich, 2012.

Schaumann, P., Kirsch, T., Kressin, H., Full scale fire test and numerical simulation of a fin plate connection protected by intumescent coating, Proceedings of 10th International Conference ASCCS, Singapore, 2012a.

Wang W.-Y., Lia G.-Q., Dong Y.-L., Experimental study and spring-component modeling of extended end-plate joints in fire, Journal of Constructional Steel Research 63 (2007) 1127–1137, 2007.

Wilson, W. M., Moore, H. F., Tests to Determine the rigidity of Riveted Joints in Steel structures, University of Illinois, Engineering Experimentation Station, Urbana, 1917.

Yu H., Burgess I. W., Davison J. B., Plank R. J., Numerical simulation of bolted steel connections in fire using explicit dynamic analysis, JConstrStRes 64 (2008) 515–525, Elsevier, 2008.