NUMERICAL STUDY OF STEEL BEAMS IN SUB-FRAME ASSEMBLY
Validation of Existing Hand Calculation Procedures

Naveed Iqbal [a], Tim Heistermann [a], Milan Veljkovic [a], Fernanda Lopes [b], Aldina Santiago [b], Luis Simões da Silva [b]

[a] Luleå University of Technology, Division of Structural and Construction Engineering, Luleå, Sweden
[b] University of Coimbra, Department of Civil Engineering, Coimbra, Portugal

Abstract
The design methods currently proposed by the codes prescribe the strength assessment of structures to be based on their strength limit state. These design methods can be applied to isolated steel members to determine their design strength in fire. The real response of a structural member is, however, more complex due to the thermal expansion and the presence of restraints against this expansion by the surrounding structure. It is therefore imperative to study the response of a structural member at high temperature in a way which includes its interaction with its surroundings. This paper will focus on the numerical investigation of steel beams in structural frames connected to concrete filled tubular (CFT) columns through reverse channel connections and comparison to hand calculation procedures.

Keywords: structural fire design, Abaqus, sub-frames, thermal expansion, catenary action, runaway deflection, artificial damping

INTRODUCTION

The present design codes prescribe that the fire resistance of structural members should be based on their critical temperature or their strength limit state (Eurocode-3, 2004). These conventional design procedures are based on the assumption that structural members are essentially isolated in their response to fire load. In reality, however, they form a part of a structural frame and their response depends heavily on the way they interact with the surrounding structure. Fire tests conducted on full scale framed buildings at the test facility in Cardington, UK have demonstrated that for the same load levels as a standard fire test, the restrained steel beams can exhibit extensive deflections and still not undergo instability (Kirby, 1997). It has been demonstrated through fire tests on sub-frames and through finite element simulations that beams can exert significant axial forces on the surrounding structure through the connections (Dai et al, 2010).
Temperature dependent variation in the axial force and vertical deflection of a restrained steel beam are important design parameters. Laboratory fire tests are very expensive and time consuming whereas Finite element modelling can be a rather complicated approach (Yin and Wang, 2005). Simplified design procedures have been proposed in order to reduce the complications and make a useful design tool, (Yin and Wang, 2005) and (Dwaikat and Kodur, 2011). The purpose of this paper is to use finite element modelling to validate the proposed hand calculation procedures by comparison to the finite element results. The finite element models in this study are sub-frame models created using the commercial software Abaqus.

1 FINITE ELEMENT MODEL (FEM)

The sub-frame models consist of a single I-Profile beam supported by two concrete filled tubular columns. The connection between the beam and columns consist of a reverse channel shown in Fig. 1, which has been shown to have greater rotational capacity at elevated temperatures (Heistermann et al, 2011). Since the test setup is symmetrical about the vertical axis through the mid-span of the beam, only half the setup has been modelled in order to save computation time.

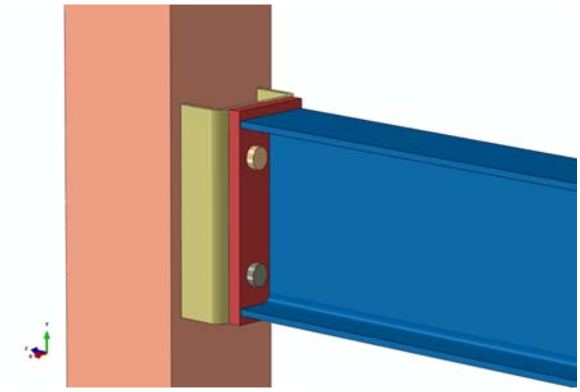

Fig. 1 Reverse channel connection

1.1 Material Properties

An elasto-plastic stress vs. strain model without hardening for steel S355 at elevated temperature has been adopted for all steel parts of the FE model (Eurocode-3, 2004). Temperature reduction factors are used to input the temperature dependent material properties. Thermal expansion has also been incorporated into the material model. Following equations relate nominal stresses and nominal strains to true stresses and true strains in Abaqus (Abaqus, 2012).

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

$$\sigma_{true} = \sigma_{nom}(1+\varepsilon_{nom}) \tag{2}$$

1.2 Contact Interactions

Application of small contact pressure for initializing contact between the contact surfaces can prevent any problems of convergence during the analysis. Small bolt load using the 'Adjust bolt length' option is applied for this purpose.

1.3 Element type

The FE model has been created using solid (continuum) element C3D8R, which is a first order reduced integration 8 node brick element. Reduced integration elements use lower order integration to calculate element stiffness matrix, which reduces the computation time. The drawback of first order reduced integration element like C3D8R is that they are prone to 'Hourglassing'. However, in Abaqus first order reduced integration elements have 'hourglass controls', which if used with finer mesh can solve the problem of hourglassing (Abaqus, 2012).

1.4 Numerical Procedure

A static general procedure is performed in the following steps.

- **Pretensioning of Bolts**
- **Load:** 40% of the bending moment capacity at ambient temperature as 4 point load.
- **Heat:** uniform Heat is applied as predefined field according to ISO 824 Fire curve

1.5 Boundary conditions

The top end of the column is free to translate longitudinally but restrained from lateral translations whereas the bottom end has all translations restrained. Both ends are free to rotate except about the longitudinal axis. At the free end of the beam a symmetry boundary condition along the longitudinal (z-axis) is defined to simulate the symmetry as discussed earlier.

1.6 Pseudo Damping of the Model

Covergence problems in the FE model can be taken care of by using artificial damping. The ratio of the dissipated energy to the total strain energy should be kept below 10% and also the support reaction forces should be checked against the applied load to prevent over damping of the model (Dai et al, 2010).

2 HAND CALCULATION PROCEDURE (HCM)

Yin and Wang in their study have proposed a hand calculation procedure, which aims to describe the restrained beam behaviour over the entire temperature range (Yin and Wang, 2005). The general methodology of the procedure is based on the equilibrium of the steel beam at all temperatures as shown in *Eq. (3)*.

$$M_{connection} + M_{midspan} + F\delta = M_{applied} \tag{3}$$

2.1 Deflection profile

The deflection profile of the beam is obtained from linear interpolation between deflection profiles for free rotation at supports and full rotational restraints as shown in *Eq. (4)* .

$$deflection\_profile = (1 - c_f) \times free\_rotation + c_f \times restrained\_rotation \tag{4}$$

The degree of end rotational restraint c_f is defined as the ratio of the rotational stiffness at supports to the beam bending stiffness.

2.2 Axial force

Initially, as the temperature increases, the beam expands against the axial restraints offered by the supporting columns. This restrained expansion produces axial compressive force in the beam shown by *Eq. (5)*.

$$F_T = K_a' \left(C_f \times \frac{\delta_m^{\ 2}}{L} - \alpha \cdot \Delta T \cdot L \right) \tag{5}$$

where K'_a effective axial support stiffness
C_f coefficient derived from the deflection profile of the beam
δ_m mid-span deflection due to mechanical load
L span length of the beam
α coefficient of linear thermal expansion
ΔT increase in temperature

2.2 Midspan deflection

During the elastic phase before the cross section yields, the midspan deflection of the beam is obtained from the equilibrium equation, shown in *Eq. (3),* through an iteration process. The midspan deflection is relatively small during the elastic phase but starts to increase excessively after yielding happens. The midspan deflection corresponding to maximum catenary force is obtained by the following compatibility equation (Dwaikat and Kodur, 2011).

$$\Delta_c^2 = \left[\frac{L'}{2}\right]^2 - \left[\frac{L}{2} + \delta\right]^2 \tag{6}$$

where L' deformed length of the beam
δ axial deformation of the beam

In the elasto-plastic phase the midspan deflection at any temperature can be obtained through linear interpolation as shown in *Eq. (7)*.

$$\Delta = \Delta_y + \frac{\Delta_c - \Delta_y}{T_c - T_y} \times (T - T_y) \tag{7}$$

where Δ_y midspan deflection at yield point
T_y temperature at yield point
T_c temperature at maximum catenary force

3 COMPARISON BETWEEN FEM AND HCM

The test setups are shown in detail in the Tab. 1.

Tab. 1 Test setups used for comparison between FEM and HCM

Model	Column	Beam	Connection	Length
Model01	SHS 250x8	UB 178x102 x19	UK SHS 180x42.7	2 m
Model02	SHS 250x10	IPE300	U200x90x10	5 m

3.1 Axial force and maximum deflection

Fig. 2 and Fig. 3 show axial force in the beam with changing temperature for different values of 'dissipated energy fraction' used as to induce damping in the model. The maximum compressive load is reached at approximately the same temperature, however for the model 02 the difference between finite element result and hand calculation result is more than that for model 01. The different values of 'dissipated energy fraction' give very little difference between the results. Fig. 4 shows the variation of midspan deflection with temperature for both models. It can be observed from Fig. 4 in comparison to Fig. 2 and 3, that the temperature at which the beam reaches its maximum compressive force is exactly the point where it starts to deflect rapidly. For zero damping the model fails to converge as the compressive force is reducing.

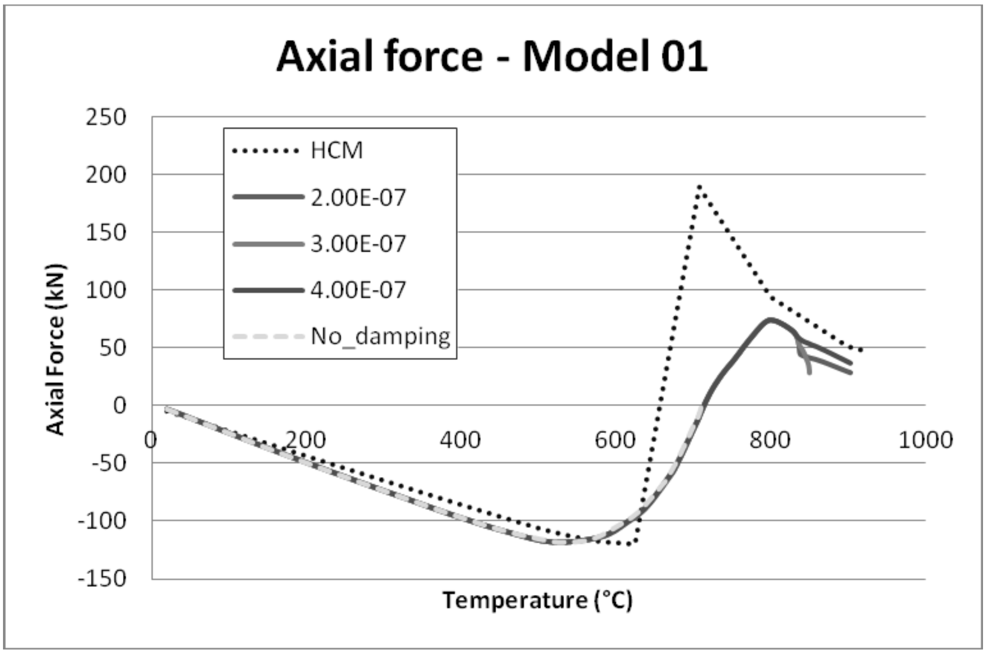

Fig 2: Axial force vs temperature for Model 01

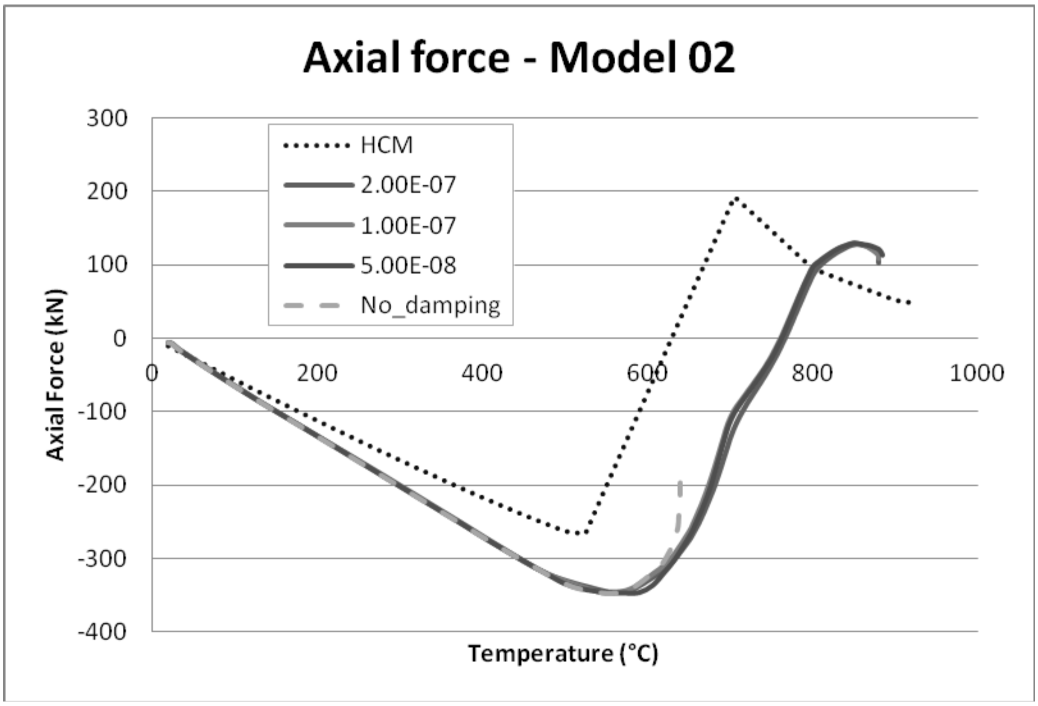

Fig 3: Axial force vs temperature for Model 02

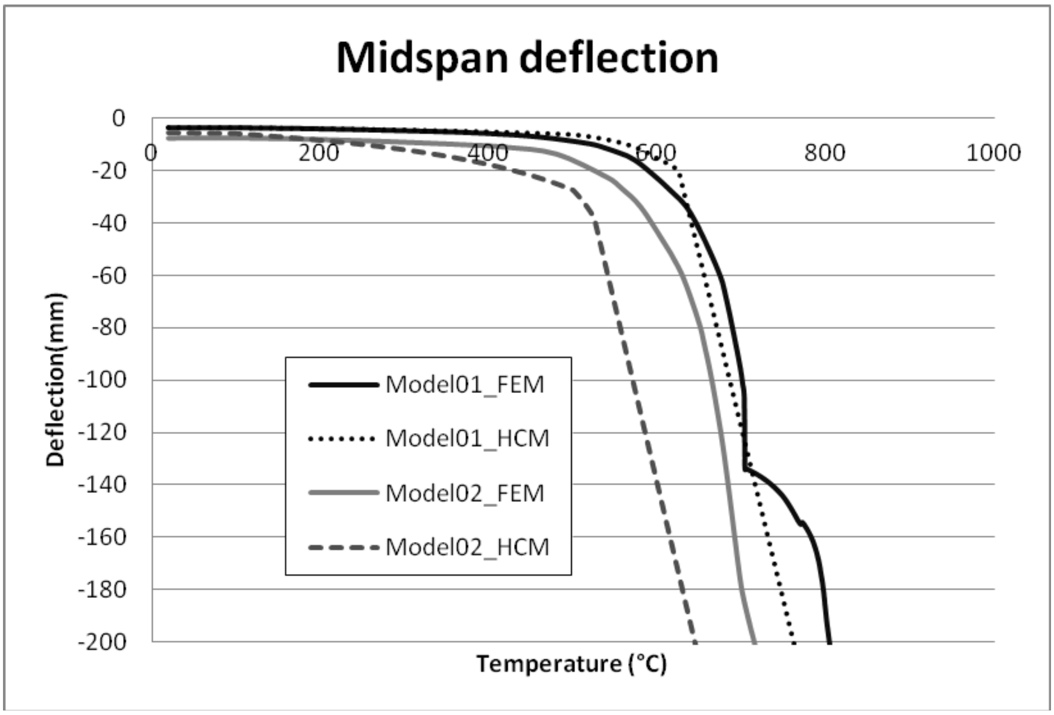

Fig 4: Midspan deflection for both models

CONCLUSIONS

Following conclusions can be drawn from the comparison between FE-modelling and the hand calculation procedure.

1. The compressive stress predicted by the hand calculation model is 0.62% lower than the result from the FE analysis for model 01 and 23% lower for model 02.
2. The hand calculation model gives conservative estimate for the maximum catenary force in the beam.
3. From hand calculation model, the limiting temperature at which the axial force is zero is lower than the FE analysis results for both models .

4. Damping of the model stabilizes the analysis and solves the convergence problem in both models.
5. The different 'dissipation energy fraction' values used here have shown to have very little impact on the results.
6. For model 02 the hand calculation produces results which are much closer to the FE analysis results.

ACKNOWLEDGMENT
The research leading to these results has received funding from the European Community's Research Fund for Coal and Steel (RFCS) under grant agreement n° RFSR-CT-2009-00021.

REFERENCES

EN 1993-1-2, Eurocode – Design of steel structures – Part 1-2: General rules, Structural fire design, CEN, European Committee for Standardization, Brussels, Belgium, 2004

Kirby B. R., Large scale fire tests: The British Steel European Collaborative Research Programme on the BRE 8 – Storey Frame, Proceedings of the Fifth International Symposium of Fire Safety Science, 1997

Dai X. H., Wang Y. C., Bailey C. G., Numerical modelling of structural fire behaviour of restrained steel beam – column assemblies using typical joint types, Engineering Structures, 2010

Yin Y. Z., Wang Y. C., Analysis of catenary action in steel beams using a simplified hand calculation method, Journal of Constructional Steel Research, 2005

Dwaikat M. M. S., Kodur V. K. R., A performance based methodology for fire design of restrained steel beams, Journal of Constructional Steel Research, 2011

Heistermann T., Iqbal N., Veljkovic M., Lopes F., Santiago A., da Silva L. S., Finite Element Modelling of Reverse Channel Connections At elevated temperatures, Proceedings of the Sixth European Conference on Steel and Composite Structures, 2011

ABAQUS, Abaqus Users' manual v6.12, Simulia, RI, USA, 2012