WP7 – GUIDELINE FOR GLOBAL STRUCTURAL ANALYSIS USING NEW BEAM-COLUMN FINITE ELEMENT

Table of contents

1 Introduction

1.1 Reminder on the background for class 4 cross-sections analysis in fire

Steel members with H or I shape class 4 cross-sections, due to their advantages regarding their lightness and efficiency, are widely used in steel constructions. However, the fire design rules of the Eurocode 3 have demonstrated to be not only very approximate but also too conservative. Additionally in the case of tapered steel members it is not clear if normal temperature design rules can be straightforward adapted for fire design.

EC3 gives simple calculation methods for fire design of class 1, 2 and 3 cross-sections in its Part 1-2 and recommends the same methods to be used with class 4 cross sections in an informative annex, suggesting that the design yield strength of steel should be taken as the 0.2% proof strength instead of the stress at 2% total strain used on the other classes of cross-sections. However, it has been demonstrated through numerical investigations, which this methodology is conservative and leads to uneconomical results. Another possibility presented in the Eurocode, is the use a very low critical temperature of 350 ºC if no calculation is performed to check the fire resistance of a class 4 member, which is even more conservative and more realistic formulae should be developed.

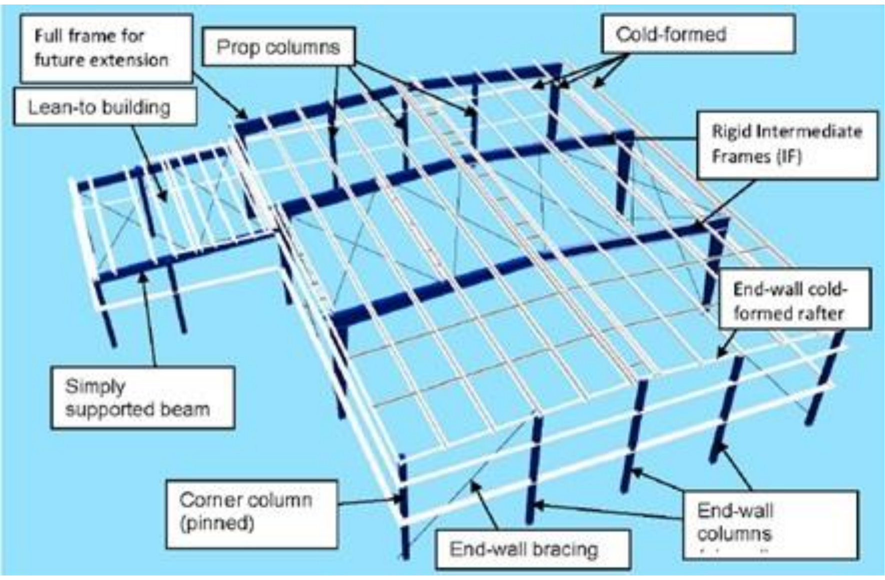

Figure 1: Typical example of primary structure (dark blue)

In the scope of this numerical computation guidance, the following aspects have been specifically developed:

- The modelling strategy to be followed by fire engineers if the structure is exposed to different “real” fire conditions

- The parameters to be considered when dealing with steel members with class 4 cross-sections in case of global structural analysis if beam-column finite elements (see) are used

- How to define the equivalent beam-column finite element.

This document has been produced as guidance and is only intended for use as such. It is not intended to provide the definitive approach in any situation, as in all circumstances. The best party placed to decide on the appropriate course of action is the engineering in charge of the considered project.

1.2 General strategy

1.2.1 Introduction

The minimum legislative level of safety for structural fire design provides an acceptable risk associated with the safety of the building occupants, fire fighters and people nearby the building.

Design for fire safety has traditionally followed prescriptive rules, but many now apply fire engineering or performance-based approaches, examples of which are given in documents EN1990 and EN1991-1-2. The fire engineering approach takes account of fire safety in its entirety, and usually provides a more fundamental and economical solution than the prescriptive approaches. Within the framework of the fire engineering approach, designing a structure involves three stages:

1. Modelling the fire scenario to determine the heat released from the fire and the resulting atmospheric temperatures within the building. (Fire Behaviour)
2. Modelling the heat transfer between the atmosphere and the structure. This involves conduction, convection and radiation, which all contribute to the rise in temperature of the structural materials during the fire. (Thermal Response)
3. Evaluating the mechanical loading under fire conditions, which differs from the maximum mechanical loading for ambient-temperature design, due to reduced partial safety factors for mechanical loading in fire. And the determination of the response of the structure at elevated temperature. (Structural Behaviour)

The design recommendations in codes contain simple checks which provide an economic and accessible procedure for the majority of buildings. For complex problems, considerable progress has been made in recent years in understanding how structures behave when heated in fires, and in developing mathematical techniques to model this behaviour, generally using the finite element method which may predict thermal and structural performance. In fire, the behaviour of the structure is more complex than at ambient temperature because changes in the material properties and thermal movements cause the structural behaviour to become non-linear and inelastic.

There are different approaches, of varying complexities, for a performance-based structural fire engineering design. The overall complexity of the design depends on the assumptions and methods adopted to predict each of three design components relating to the fire severity, heat transfer and structural response. The different steps are illustrated in the following scheme:

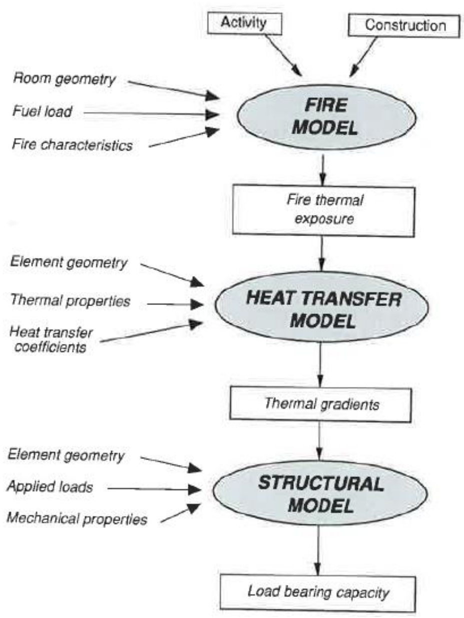

Figure 2: Steps to conduct a fire engineering design

1.2.2 Fire behaviour

1.2.2.1 Basic principles of the fire behaviour

The basic development of an enclosed uncontrolled compartment fire can be divided into a number of stages, as follows:

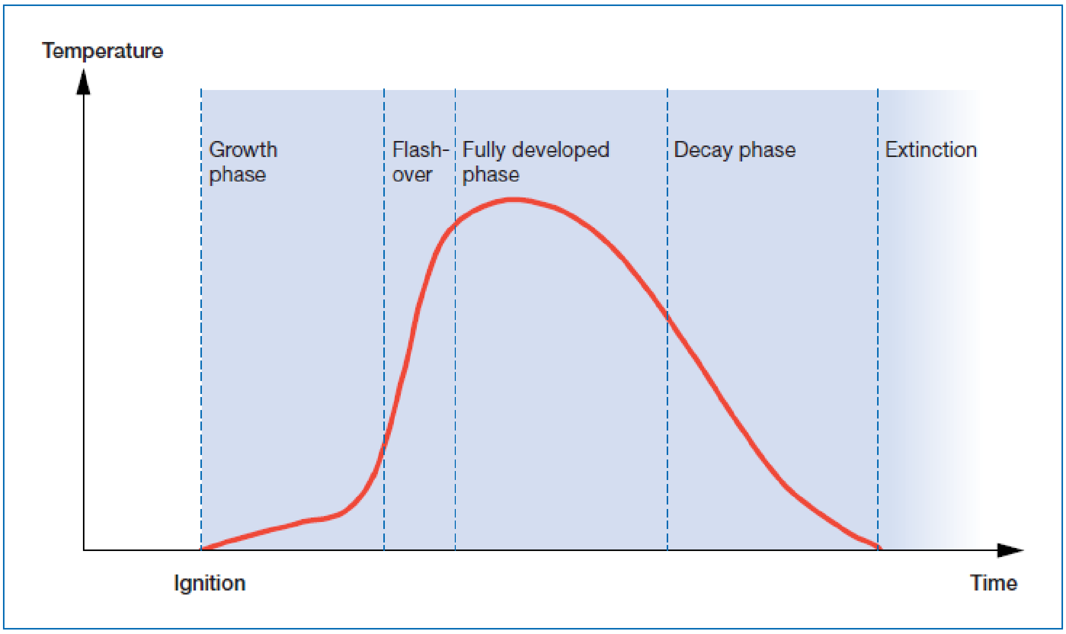

Figure 3: Evolution of the temperature in function of time for a typical “real” fire

Growth phase (pre-flashover): Ignition defines the beginning of the fire development. At the initial growth phase, the fire will normally be small and localized within the compartment and may stop at this stage. Smoke and combustion products (pyrolysis) will accumulate beneath the ceiling gradually forming a hotter upper layer in the compartment, with a relatively cooler and cleaner layer at the bottom. With sufficient supply of fuel and oxygen, and without the interruption of firefighting or other active measures, the fire will continue to grow with the release of more hot gases and pyrolysis to the smoke layer. The smoke layer will descend as it becomes thicker. If the growth of the fire is slow due to lack of oxygen or combustible material in the proximity of the fire then the fire remains localized.

Flashover: If the development of the fire leads to sufficiently hot gases in the compartment (approximately 550-600ºC), sudden ignition of all combustible objects within the compartment will occur. This phenomenon is known as flashover with the whole compartment engulfed in fire.

Fully developed phase (post-flashover): Following the flashover, the fire enters a fully developed stage with the rate of heat release reaching a maximum and the burning rate remaining substantially steady. The burning rate may be limited by availability of ventilation or fuel. Normally this is the most critical stage which, unless controlled, can lead to possible wide spread structural damage and fire spread to other compartments.

Decay phase: After a period of sustained burning, the rate of burning decreases as the combustible materials are consumed and the fire enters the decay phase.

Extinction: The fire will eventually cease when all combustible materials have been consumed and there is no more energy being released.

The thermal actions to be used in subsequent analysis may be either nominal, derived from simple calculation, or by advanced methods. The choice of a particular fire design scenario should be based on a risk assessment taking into account the likely ignition sources and any fire detection/suppression methods available. The design fire should be applied to only one fire compartment at a time.

1.2.2.2 Nominal temperature-time curves

The nominal fire curves provide a simple means of assessing building materials and components against a common set of performance criteria subject to a closely defined thermal and mechanical loading under prescribed loading and support conditions. Although notionally a representation of building fires, the standard fire curves do not take into account of any of physical parameters affecting fire growth and development. The nominal curves given in EN 1991-1-2 are described below.

- Standard temperature-time curve:

The standard fire curve has been used effectively for many years to determine the relative performance of construction materials. The temperature-time relationship is described below and set out in EN1363.

$$\theta_g = 20 + 345 log_{10}(8t + 1)$$

Where:

θ_g is the gas temperature in the fire compartment (ºC)

t is the time (min)

One crucial shortcoming of this curve and others is that there is no descending branch, no cooling phase. It has been shown that the cooling phase can be very important with regard to structural performance, particularly where large thermal restraint is present. This standard relationship is the basis for the tabulated data in the codes. Many of the design methods available through the Eurocodes are restricted to the choice of a design fire similar to the standard curve, as there is insufficient information on the thermal and structural performance of members and complete structures subject to natural fire exposures.

- External fire curve:

The external fire curve is used for structural members in a façade external to the main structure. The external fire curve is given by:

$$\theta_g = 660(1 - 0.687e^{-0.32t} - 0.313e^{-3.8t}) + 20$$

Where:

θ_g is the gas temperature in the fire compartment (ºC)

t is the time (min)

- Hydrocarbon curve:

In situations where petrochemicals or plastics form a significant part of the overall fire load, the temperature rise is very fast due to the much higher calorific values of these materials. Therefore, for such situations, an alternative temperature-time curve has been developed of the form:

$$\theta_g = 1080(1 - 0.325e^{-0.167t} - 0.675e^{-2.5t}) + 20$$

Where:

θ_g is the gas temperature in the fire compartment (ºC)

t is the time (min)

1.2.2.3 *Equivalent time of fire exposure*

EN 1991-1-2 includes a method to determine the appropriate fire resistance period for design based on the consideration of the physical characteristics of the fire compartment. This is effectively a "halfway" between the nominal curves and the behaviour of a realistic fire compartment. The method relates the severity of a real fire in a real compartment to an equivalent period of exposure in a standard test furnace. The relevant input parameters are the amount of fire load, the compartment size (floor area and height), the thermal properties of the compartment linings and the ventilation conditions.

$$t_{e,d} = (q_{f,d} k_b w_f) k_c$$

Where:

$t_{e,d}$ is the equivalent time of fire exposure for design (min)

$q_{f,d}$ is the design fire load density (MJ/m^2)

k_c is a correction factor dependent on material.

k_b is a conversion factor dependent on thermal properties of linings

w_f is the ventilation factor

1.2.2.4 *Parametric temperature-time curves*

Along with the time equivalent approach, parametric fires are an example of the simple calculation methods to determine the compartment internal atmosphere time-temperature relationship. The basic formulation is based on the work carried out by Wickström. The parametric approach provides a quick and easy approximation of compartment gas temperatures ideally suited for use on modern spreadsheets. The approach has been extensively validated over a number of years. It applies only to the post-flashover phase, which is of primary concern when considering structural issues and assumes a uniform temperature within the compartment. The basic formulation in Annex A of EN 1991-1-2 is as follows:

$$\theta_g = 20 + 1325(1 - 0.324e^{-0.2t^*} - 0.204e^{-1.7t^*} - 0.472e^{-19t^*})$$

Where:

θ_g is the temperature in the fire compartment (ºC)

$t^* = t\Gamma$

t is time (h)

$\Gamma = [O/b]^2 /(0.04/1160)^2$

$\mathrm{b} = \sqrt{(\rho c \lambda)}\ (\frac{J}{m^2 s^{1/2} K})$

$O = \left(\frac{A_v \sqrt{h}}{A_t}\right) (m^{\frac{1}{2}})$ opening factor

A_v is the area of vertical openings (m^2)

h is the height of vertical openings (m)

A_t is the total area of enclosure (m^2)

ρ is the density of boundary enclosure (kg/m^3)

c is the specific heat of boundary of enclosure (J/kgK)

λ is the thermal conductivity of boundary (W/mK)

1.2.2.5 Advanced fire models

In certain circumstances it may be necessary to go beyond a reliance on nominal fire exposures or simple calculation methods. Advanced methods, including zone models based on a solution of equations for conservation of mass and energy or more complex computational fluid dynamics (CFD) models, may be used to provide information based on a solution of the thermodynamic and aerodynamic variables at various points within the control zone. Such models have been used effectively for many years to model the movement of smoke and toxic gases and are now being extended to model the thermal environment for particular post-flashover fire scenarios.

1.2.2.5.1 Zone models

Zone models divide an enclosure into a small number of distinct regions, each of which is characterized by a set of time dependent variables that describe its physical state. Each zone is considered isothermal and homogeneous.

Conservation equations for mass and energy are applied to each zone so that the relationships between the physically significant parameters and their evolution can be determined. Whilst zone models are more primitive in conception than field models, they are easier to apply and provide far more economic results in terms of computational requirements. They are the most practical method for achieving first order approximations to real fire behaviour.

Zone models assume a hot upper layer (or zone) and a cool lower layer (or zone). Interaction between the two zones takes place through the fire plume above the burning object. The fire plume rises through buoyancy to the ceiling, entraining cool air as it rises. The combustion products and entrained air then spread across the ceiling. Once the walls are reached, the hot layer increases in thickness until such time as its depth is controlled by the ventilation through the openings. The fire stabilizes its burning rate to match the available air supply.

If there are no large openings the hot layer will descend to the level of the fire, and the rate of combustion will drop as the fire is starved of oxygen.

There are number of zone models that are or have been available to users. Examples include the FIRE SIMULATOR single room model in FPETool (Nelson, 1990; Deal, 1993), the CFAST family of models (Peacock et al, 1997; Jones et al, 1999) and Ozone from the University of Liege in Belgium. The following schema illustrates the principles of a zone model:

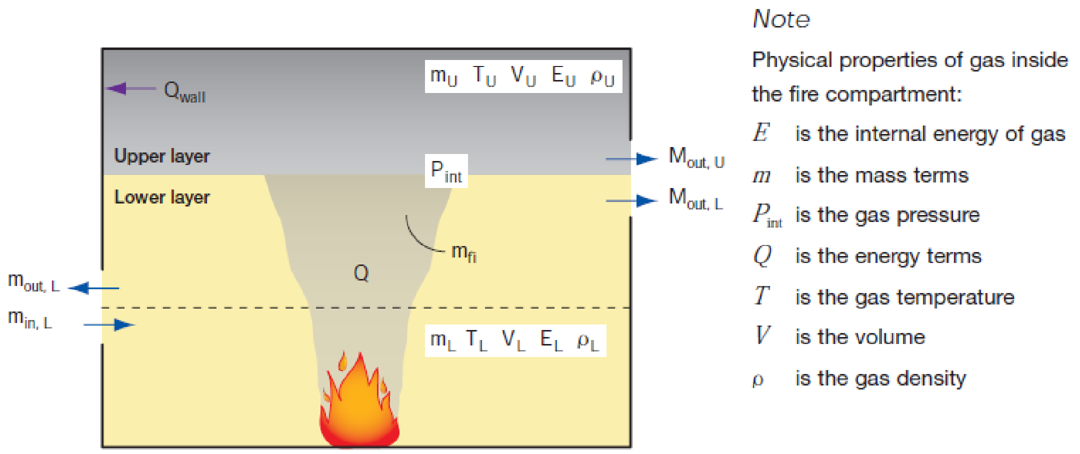

Figure 4: Zone model illustration

1.2.2.5.2 Field models

In field modelling, the compartments are divided into many thousands of computational cells throughout the enclosure. Field models, often called Computational Fluid Dynamics (CFD) models solve the conservation mass momentum, energy and species in each cell, thus giving a three dimensional field of the dependent variables including temperature, velocity, concentration, etc. Field models use complex fluid mechanics algorithms such as the K-E method to solve for the turbulent flow present in fires. The K-E method is a time averaging technique, which smears out much of the complex flow details that are sometimes required for detailed flow modelling. An alternative technique is to use the Large Eddy Simulation technique to deal with unsolved, sub-grid scale motion.

Field models require a great deal of experience on the part of the user and place great demands on computational facilities. Field models have been developed to stage where they can be applied for design purposes although there is still much research required and so must be used with great care.

A consequent number of fire field models are currently available. They fall into one of two camps, specific fire field modelling software that is intended only for modelling fire and general purpose CFD codes that can be used for fire modelling applications. The general purpose CFD codes PHOENICS, CFX and Fluent are examples of the latter. These general purpose CFD products are often used in fire modelling applications.

The fire field models, JASMINE (developed by FRS and which makes use of an early version of the PHOENCIS code as its CFD engine), KAMELON (developed by SINTEF/NTH), SMARTFIRE (developed by UoG) and SOFIE (developed by Cranfield/FRS) are examples of the former.

And, finally, Fire Dynamics Simulator (FDS) is a well-known large-eddy simulation (LES) code for low-speed flows, with an emphasis on smoke and heat transport from fires and is continuously developed by NIST (National Institute of Standards and Technology).

1.2.3 Thermal response

1.2.3.1 Basic principles of heat transfert

Heat transfer is the evaluation of the energy transfer that takes place between material bodies as a result of a temperature gradient. The three modes of heat transfer are conduction, convection and radiation.

Heat transfer analysis is undertaken to determine the temperature rise and the distribution of temperature within the structural members. Thermal models are based on acknowledged principles and assumptions of heat transfer. Thermal models vary in complexity ranging from simple tabulated values to complex calculation models based on finite difference or computational fluid dynamics. The heating conditions considered extend to cover natural fire scenarios. However, the validity of some of the simple methods and most of the tabular data is restricted to a fire exposure corresponding to the standard fire curve.

In order to undertake this analysis, knowledge of steel properties at elevated temperature is required, specifically: thermal conductivity, specific heat, density and emissivity.

The surface of a structural member exposed to a fire is subject to heat transfer by convection and radiation. Typically, the radiation is more dominant than the convection except for the very early stages of the fire.

The thermal actions can be represented by the net heat flux h_{net} given by the following equations:

$$h_{net} = h_{net,c} + h_{net,r}$$

Where:

$h_{net,c} = \alpha_c(\Theta_g - \Theta_m)$ is the net heat flux due to convection

$h_{net,r} = \Phi \varepsilon_f \varepsilon_m [(\Theta_r + 273)^4 - (\Theta_m + 273)^4]$ is the net heat flux due to radiation

Where:

α_c is the coefficient of heat transfer by convection in $W/m^2.K$. Some typical values are given in the following table for different fires:

Fire model	**α_c (W/m^2K)**
Standard fires	25
External fires	25
Hydrocarbon fires	50
Parametric fires	35
Unexposed side of separating members: **- Without radiation** **- With radiation**	 4 9

Table 1: Typical α_c values

Θ_g is the gas temperature in the vicinity of the fire exposed member (ºC)

Θ_m is the surface temperature of the member (ºC)

ε_f is the emissivity of the fire (=1.0)

ε_m is the surface emissivity of the member depending on material (carbon steel => 0.7)

Φ is the configuration factor (<=1.0). The annex G of EC1 provides guidance to calculating the value of Φ. It can be taken as 1 in a conservative way.

Θ_r is the effective radiation temperature of the fire environment (ºC)

σ is the Stephan-Boltzmann constant (5.67×10^{-8} $W/m^2.K^4$)

1.2.3.2 Simplified calculation method models for structural steel

The European fire design standard for steel structures includes methods for calculating the temperature rise in both unprotected and protected steel, assuming a uniform temperature distribution through the cross section and based on a lumped mass model.

The rise in temperature is given by:

$$\Delta\theta_{a,t} = k_{sh}\frac{A_m/V}{c_a\rho_a}h_{net,d}\Delta t \quad for\ \Delta t \leq 5sec$$

Where:

ρ_a is the unit mass of steel (kg/m^3)

c_a is the specific heat of steel (J/kg.K)

$h_{net,d}$ is the net heat flux per unit area (W/m^2)

A_m is the surface area of the member per unit length (m^2/m)

V is the volume of the emember per unit of length (m^3/m)

A_m/V is the section factor for unprotected steel members (m^{-1})

Δt is time interval (s)

k_{sh} is the correction factor for the shadow effect

The k_{sh} correction for "shadow effects" accounts for the fact that members with geometry similar to I and H sections are shielded from the direct impact of the fire in some parts of the surface. For circular or rectangular cross-sections fully engulfed by fire the shadow effect is not relevant and k_{sh} = 1 otherwise:

$$k_{sh} = \frac{0.9\left[A_m/V\right]_b}{A_m/V} \text{ for I sections under nominal fire actions}$$

$$k_{sh} = \frac{\left[A_m/V\right]_b}{A_m/V} \text{ for other cases}$$

In above equation the value of A_m/V should not be used if it is less than 10 m^{-1}. $\left[A_m/V\right]_b$ is the box value of the section factor.

For protected members, a similar procedure is adopted taking into account the relevant material properties of protection material. The method is applicable to non-reactive fire protection systems such as board or spray protection but is not appropriate for reactive materials such as intumescent coatings. Assuming a uniform temperature distribution, the temperature rise $\Delta\theta_{a,t}$ of a protected steel member during a time interval Δt is given by:

$$\Delta\theta_{a,t} = \frac{\lambda_p\, A_p/V}{d_p c_a \rho_a}\ \frac{(\theta_{g,t} - \theta_{a,t})}{(1-\phi/3)}\Delta t - \left(e^{\phi/10} - 1\right)\Delta\theta_{g,t}$$

With:

$$\Delta\theta_{a,t} \geq 0 \; and \; \phi = \frac{c_p \rho_p}{c_a \rho_a} d_p A_p / V$$

λ_p is the thermal conductivity of fire protection material (W/m.K)

$\theta_{a,t}$ is the steel temperature at time t (ºC)

$\theta_{g,t}$ is the ambient gas temperature at time t (ºC)

A_p/V is the section factor for steel members insulated by fire protection material (m^{-1})

A_p is the appropriate area of fire protection material per unit length (m^2)

d_p is the thickness of fire protection material (m)

c_a is the temperature dependent specific heat of steel (J/kg.K)

ρ_a is the unit mass of steel (kg/m^3)

c_p is the temperature independent specific heat of fire protection material (J/kg.K)

ρ_p is the unit mass of fire protection material (kg/m^3)

Δt is time interval (s)

1.2.3.3 Advanced numerical models

Advanced models for heat transfer problems require adapted computer software. Heat transfer is a transient-state condition, coupled with time-dependent boundary conditions and temperature-dependent material properties. Consequently, most advanced models can only be developed based on finite difference or finite element techniques. The heat transfer analysis can be performed using a two-dimensional or a three dimensional model. The general aspects to take into account for the modelling of heat transfer analysis are the following ones:

Meshing: The shape and dimensions of the structural model are modelled by a finite element mesh of general flow continuum elements, in the form of triangles, quadrilaterals, wedges, or bricks. The boundary elements or interface elements can be line shaped elements for a 2-D model, and triangular or quadrilateral elements for a 3-D model.

Boundary Conditions: Heat sources can be represented by either temperature-time functions or heat flux in boundary elements. Convection and/or radiation at boundaries of the structural model can be modelled by the heat transfer coefficient of boundary elements.

Material properties: The material can be isotropic, orthotropic or anisotropic.

The material thermal properties of conductivity, specific heat and emissivity can be temperature-dependent.

1.3 Structural fire engineering

1.3.1 Introduction

The simplest method to predict the structural behaviour of buildings in fire is to analyse individual members at the fire limit state using partial load and material safety factors, which take into account realistic loads at the time of the fire and actual material strengths. These methods are given in the codes and design guides and take into account the reduction in strength and stiffness of materials during a fire. Simple design methods, which are based on fundamental engineering principles, can be used irrespective of the fire model used. However, some empirical structural design methods are only valid for use with the standard time-temperature fire model, which was used in their derivation.

Simple plastic design methods exist to consider frame behaviour in a fire. In the Eurocodes, frame behaviour is used to allow the effective lengths of continuous steel columns to be reduced from ambient temperature values.

The simple design models for individual members and sub-frames are assumed to be conservative but do ignore some aspects of the actual behaviour of real buildings. A possible design approach to predict more accurately the behaviour of buildings in fire is to use finite element analysis. The approach incorporates the stress-strain-temperature relationship of materials and can predict stresses and deformations throughout the whole structure. Expertise is required to use these advanced models and special care is required in defining the types of elements used, boundary conditions, localised behaviour and interpretation of the results.

Finite element modelling of whole building behaviour can provide a more accurate estimation and understanding of the structural response, over the full duration of the defined fire, compared to other methods.

The overall frame stability in a fire should be considered. For braced frames no additional checks are normally required provided a sufficient number of cores or bracing, that provide the lateral resistance, have adequate fire resistance, shielding or containment within fire resisting cores. For sway frames, a frame analysis at elevated temperatures is required to ensure sufficient overall stability during a fire.

The available structural design methods are summarised in the following table:

Model	**Simple element**	**Sub-models**	**Advanced computer finite-element models**
Complexity	Simple	Intermediate	Advanced
Input parameters	• Temperature through the cross-section • Material strength and stiffness reduction • Applied static load • Simplified boundary conditions	• Temperature through the cross-section and along the member • Material strength and stiffness reduction • Applied static load • Boundary conditions	• Temperature through and along the cross-section • Full material stress-strain-temperature relationship • Applied static load • Boundary conditions • Element type and density
Accuracy	• Ignores real behaviour but assumed to be conservative • Ultimate strength calculation	• Begins to consider actual load paths and restraint • Ultimate strength calculation	• Predicts internal stresses, displacements, and rotations for all members throughout the duration of the fire • Localised behaviour is not modelled accurately in whole building modelling
Design tools	• Simple equations for hand calculations	• Simple equations for hand calculations • Plastic design, redistribution of moments • Simple computer models	• Commercially available or purpose written computer software

Figure 5: Available structural design

It is worth emphasising that the analysis of the structure will only be as accurate as the fire modelling and thermal analysis. Therefore the accuracy of all three components of the design should be considered when assessing the final analysis.

1.3.2 Basic principles

All materials lose strength and stiffness at elevated temperatures. Based on test results, design codes present simplified values of strength and initial stiffness for various materials at different temperatures.

The codes also provide simplified stress-strain temperature relationships for steel and concrete, which can be used within advanced models.

At the same time, all materials will expand, to some extent, when heated. If a non-uniform temperature distribution forms through the section, thermal curvature will occur with the element generally deflecting towards the heat source. Any resistance to the free movement of axial thermal expansion or thermal curvature will induce internal stresses within the member. In addition, due to assuming plane sections remain plane, any non-linear temperature distribution through an element will induce internal thermal stresses.

1.3.3 Simple calculation methods

The simplest calculation methods are based on the behaviour of individual members. These members could be in the form of a column, beam, wall or floor slab. Guidance on the design of structural members is presented in codes and design guides.

With member design, the effects of restraint to axial thermal expansion are ignored. However, the effects of thermal gradients through the cross section are generally considered.

The simple member calculation methods are typically based on strength and provide no detail on the displacement history, or maximum displacement, of the member during the fire.

If the design approaches are based on fundamental engineering principles, with the strength of materials within the member being reduced with increase in temperature, then they are valid for any fire scenario.

However, there are some cases where the design procedures given in the codes (particularly relating to composite construction and timber members) are only valid for the standard time-temperature fire scenario, since they have been derived from, and validated against, standard fire test results. The designer should check that the calculation approach adopted for estimating the structural response is valid for the fire scenario considered.

It is generally accepted that the available calculation methods for the design of individual members will provide acceptable conservative answers. However, the design approach ignores the true structural response of the building, which can be either detrimental or beneficial to the survival of the building as a whole.

The important modes of behaviour that are generally ignored in member design are described below:

- The effects of thermal expansion of the beams laterally displacing external columns.
- Any induced forces acting on a wall due to the movement of the heated structure in the proximity of the wall.
- The effect of induced compressive forces due to restrained thermal expansion. These induced compressive forces could cause buckling of vertical elements, local buckling of beams, or increase the beneficial effect of compressive membrane action.
- Re-distribution of moments with frame action.
- Any pulling in of external columns from catenary action of beams.
- Any beneficial effect of alternative load paths, catenary action or membrane action.

Consideration should be given to these modes of behaviour when detailing members and connections.

Simple design methods to determine load-bearing capacity are available for steel tension, compression and beam members. If a uniform temperature distribution is assumed through the member then the calculation is simply based on a reduction in yield strength. Simple calculation methods are available to take into account varying temperature distribution through the member and along the member's length.

The design of steel members is based on the engineering principles applied in the normal cold design except that the effects of reduction in material strength and stiffness are taken into account, together with partial safety factors that relate to the fire limit state.

1.4 Current issue with class 4 cross-section design

It is well known that if the walls of steel members have high slenderness, some local instability called as local buckling will occur which could reduce largely the overall loadbearing capacity of steel members. In order to take account of such a phenomenon in room temperature design of steel members, the part 1.1 of Eurocode 3 has recommended to classify steel members according to their wall slenderness and the loading condition to four different classes which means that the higher the class of steel member is, the more important the influence of local buckling over the global loadbearing capacity becomes.

The above design concept has been also introduced into the fire part of Eurocode 3 for fire design of steel members. However, for a long time, the fire part of Eurocode 3 deals with the fire resistance of steel members composed of very thin walls so with high risks of local buckling, named as class 4 cross-section steel members by Eurocode, using an extremely simple rule based on a unique critical temperature of 350 °C whatever their load ratio is.

Obviously, this rule could lead to a not necessary high cost of fire protection if the steel members with local buckling risks are subjected to low load ratio. As a comparison, steel members without local buckling risk have a critical temperature of at least 500 °C which could be even higher if their load ratio is lower than 0.65. With this respect, in the new version of Eurocode 3 (EN 1993-1-2), it has been incorporated in its informative annex E a simple calculation method trying to provide a design tool for fire engineers to get more detailed results for the fire resistance of such type of steel members. Apparently, for the purpose of simplicity, the calculation methods recommended in this annex for the overall load bearing capacity of class 4 cross-section steel members are referred to those given in the main part of EN 1993-1-2 for steel members with cross-section class lower than 4 considered as without local buckling risks in fire situation. It had been shown that these simple calculation rules could lead to very conservative fire resistance of such type of steel members, especially in case of high load ratio around 0.7. For example, a failure temperature of 26 °C is even predicted by the simple calculation rules which are unfortunately fully unrealistic.

To take into account the effect of local buckling that can occur in slender plates or plated structures subjected to compressive in-plane loading, Part 1-5 of Eurocode 3 presents two different calculation methods: the effective width method and the reduced stress method. The former is strongly efficient for standard geometries, being the resistance of plated members determined using the effective areas of plate elements in compression for class 4 sections using cross sectional data (A_{eff}, I_{eff}, W_{eff}) for cross sectional verifications and member verifications for column buckling and lateral torsional buckling according to EN 1993-1-1. However, the effective width method is not applicable for non-uniform geometries and certain types of loading. On the contrary, the reduced stress method can be applied to almost any geometry and loading due to the generic concept that takes into account the full stress field and its interaction, as mentioned in section 10 of EN 1993-1-5.

Although some studies have been done previously within the scope of one research project for welded or hot-rolled class 4 steel members this type of study is very limited and cover only, for example the buckling of class 4 steel columns or are related to other types of steel, for example stainless steels which constitutive law are different from carbon steel.

2 Modelling strategy for class 4 cross-section steel structures

2.1 Computer numerical analysis and finite element method

2.1.1 Basics

Today the finite element method (FEM) is considered as one of the well-established and convenient technique for the computer solution of complex problems in different fields of engineering: civil engineering, mechanical engineering, nuclear engineering, biomedical engineering, hydrodynamics, heat conduction, geo-mechanics, etc. From other side, FEM can be examined as a powerful tool for the approximate solution of differential equations describing different physical processes.

The success of FEM is based largely on the basic finite element procedures used: the formulation of the problem in variation form, the finite element discretization of this formulation and the effective solution of the resulting finite element equations. These basic steps are the same whichever problem is considered and together with the use of the digital computer present a quite natural approach to engineering analysis.

A complete computer numerical analysis usually consists of three distinct stages: pre-processing, simulation, and post-processing. These three stages are linked together by files as shown below:

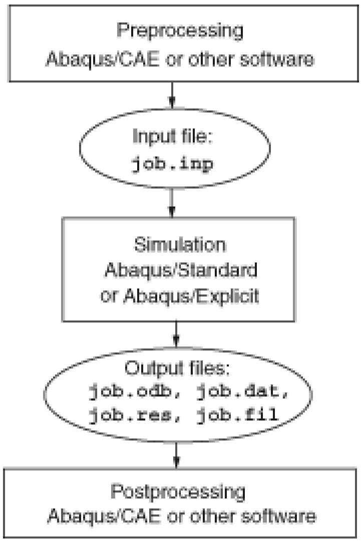

Figure 6: Steps for a FEM analysis

Pre-processing

In this stage you must define the model of the physical problem and create an input file. The model is usually created graphically using a pre-processor, although the input file for a simple analysis can be usually created directly using a text editor.

Simulation or solver

The simulation, which normally is run as a background process, is the stage in which the code solves the numerical problem defined in the model. Examples of output from a stress analysis include displacements and stresses that are stored in binary files ready for post-processing. Depending on the complexity of the problem being analysed and the power of the computer being used, it may take anywhere from seconds to days to complete an analysis run.

Post-processing

The designer can evaluate the results once the simulation has been completed and the displacements, stresses, or other fundamental variables have been calculated. The evaluation is generally done interactively using a postprocessor, which reads the neutral binary output database file, it normally has a variety of options for displaying the results, including colour contour plots, animations, deformed shape plots, and *X–Y* plots.

2.1.2 Components of a numerical model

The model is composed of several different components that together describe the physical problem to be analysed and the results to be obtained. At a minimum the analysis model consists of the following information: discretized geometry, element section properties, material data, loads and boundary conditions, analysis type, and output requests.

Discretized geometry

The structure is transferred into a discrete system by dividing (meshing) the structure into finite elements. Finite elements and nodes define the basic geometry of the physical structure being modelled. Each element in the model represents a discrete portion of the physical structure, which is, in turn, represented by many interconnected elements. Elements are connected to one another by shared nodes. The collection of all the elements and nodes in a model is called the *mesh*. Generally, the mesh will be only an approximation of the actual geometry of the structure.

Five aspects of an element characterize its behaviour: Family, Degrees of freedom (directly related to the element family), Number of nodes, Formulation and Integration. The element type, shape, and location, as well as the overall number of elements used in the mesh, affect the results obtained from a simulation. As the mesh density increases, the analysis results converge to a unique solution, and the computer time required for the analysis increases. A balance needs to be made between the number of elements used and the required accuracy. This can only be assessed by carrying out a sensitivity analysis which involves conducting the same structural analysis but increasing the number of finite elements used. The solution obtained from the numerical model is generally an approximation to the solution of the physical problem being simulated. The extent of the approximations made in the model's geometry, material behaviour, boundary conditions, and loading determines how well the numerical simulation matches the physical problem.

Connecting the finite elements together at nodal points needs careful consideration. It has been shown that the behaviour of structures during fire is predominantly governed by restraint to thermal expansion. It is therefore important that the elements are connected at the correct points to ensure accurate representation of thermal restraint.

Element section properties

The different commercial finite element solvers have a wide range of elements, many of which have geometry not defined completely by the coordinates of their nodes. For example, the layers of a composite shell or the dimensions of an I-beam section are not defined by the nodes of the element. Such additional geometric data are defined as physical properties of the element and are necessary to define the model geometry completely. The type of finite element used to model the structure needs to be defined. The following guidance is offered:

- Beam-column elements are line elements, modelling one-dimensional stress state, which include axial and flexural terms. They can be used effectively to model columns and beams. Integrating across the cross section at several points along the element allows any cross-sectional variation to be included. It is important to ensure that the numerical integration across the cross-section accurately models any variation in material and temperature.
- Spring elements are elements used to represent the variation of stiffness and strength between two nodal points that are in close proximity. These elements can be used to model connections.
- Shell elements are planar elements, modelling two-dimensional stress state, which include both membrane and flexural terms. Integrating through the thickness of the element allows the variation of the properties to be included. These elements are typically used to model floor

slabs and the class 4 cross sections, which are given to have local buckling in the section before they get the yield point.

Material data

Material properties for all elements must be specified. While high-quality material data are often difficult to obtain, particularly for the more complex material models, the validity of the results is limited by the accuracy and extent of the material data.

For the one-dimensional stress state, the stress strain- temperature relationship given in the codes can be used for steel. For the two-dimensional stress state a biaxial stress-strain-temperature relationship should be used. Strain reversal during both the heating and cooling stage of the fire should be considered, if it is detrimental to the structural behaviour and always when natural heating action has been defined as a thermal action.

Loads and boundary conditions

Loads distort the physical structure and, thus, create stress in it. The most common forms of loading include:

- point loads
- pressure loads on surfaces
- distributed tractions on surfaces
- distributed edge loads and moments on shell edges
- body forces, such as the force of gravity
- thermal loads

The applied static load should comply with the codes assuming fire limit state design. The rise in temperature, together with accurate thermal gradients should be applied in discrete steps to avoid numerical instability. The range of design fires encompassing low temperature maximum duration and high temperature minimum duration should be considered to identify the worst case in terms of structural response.

Boundary conditions are used to constrain portions of the model to remain fixed (zero displacements) or to move by a prescribed amount (nonzero displacements).

The boundary conditions should be defined. Due to the effects of restrained thermal expansion, the definition of boundary conditions can be important. It may be found that the slightest variation in boundary conditions results in significant changes in the estimated response. Boundary conditions can fall into two categories. The first relates to actual boundaries of the structure, which are fairly easy to define. The second relates to boundaries of a sub-model where the fixity at the boundary represents the rest of the structure which is not actually modelled. If it is found that variations of fixity have a significant effect on the predicted behaviour using a sub-model then the modelled area should increase and the boundary be moved away from the modelled area of interest.

Initial geometric imperfections should be applied to the columns and any laterally unrestrained beams. An initial imperfection of span/1000 is generally adequate.

In a static analysis enough boundary conditions must be used to prevent the model from moving as a rigid body in any direction; otherwise, unrestrained rigid body motion causes the stiffness matrix to be singular. A solver problem will occur during the solution stage and may cause the simulation to stop prematurely. Usually, it will issue a warning message if it detects a solver problem during a simulation. It is important that you learn to interpret such error messages. If you see a “numerical singularity” or “zero pivot” warning message during a static stress analysis, you should check whether all or part of your model lacks constraints against rigid body translations or rotations. Rigid body motions can consist of both translations and rotations of the components. The potential rigid body motions depend on the dimensionality of the model.

In dynamic analysis inertia forces prevent the model from undergoing infinite motion instantaneously as long as all separate parts in the model have some mass; therefore, solver problem warnings in a dynamic analysis usually indicate some other modelling problem, such as excessive plasticity.

Analysis type

The well-known FEM programs can carry out many different types of simulations, but this guide only covers the two most common: static and dynamic stress analyses.

In a static analysis the long-term response of the structure to the applied loads is obtained. In other cases the dynamic response of a structure to the loads may be of interest: for example, the effect of a sudden load on a component, such as occurs during an impact, or the response of a building in an earthquake.

Output requests

The simulation can generate a large amount of output. To avoid using excessive disk space, the designer can limit the output to that required for interpreting the results.

Apart from the essential components of a finite element model, it is important to keep in mind that Finite-element analysis is a design tool to estimate the structural response. Similar to other design methods, assumptions and approximations are embedded within the method.

When using finite element models to predict the structural response of a building to a given defined temperature distribution, a sensitivity assessment may be required to assess the effect of mesh density, connection behaviour and boundary conditions adopted for sub-models.

2.2 Parameters to take into account in the modelling process and their influence on class 4 cross-section

2.2.1 Hot-rolled sections and welded sections (constant and tapered)

Shell finite elements were used to model the class 4 steel cross section columns and beams in order to take into account the local buckling of the different plates, flanges and web, which are suitable to suffer this type of instability:

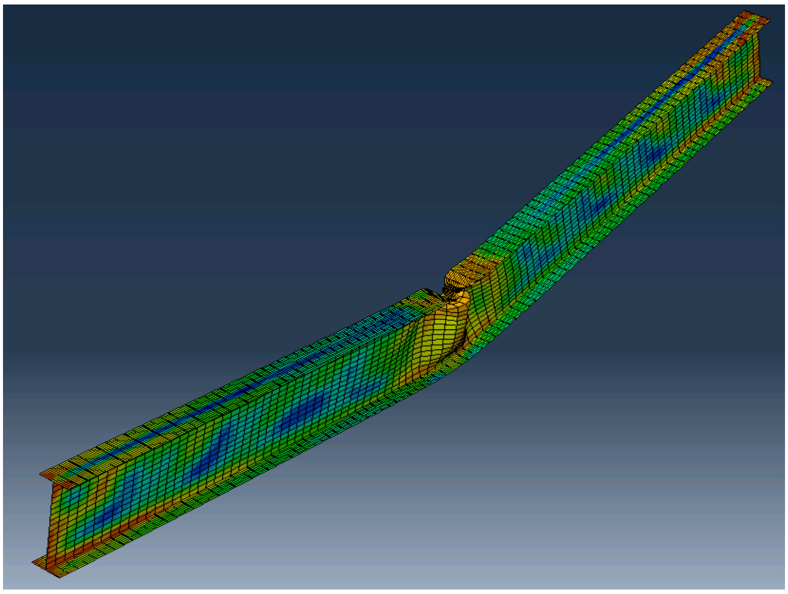

Figure 7: Illustration of a beam local instability

The single structural elements (beams or columns) were meshed using quadrilateral conventional shell elements (namely type S4). Conventional shell elements discretize a body by defining the geometry at a reference surface. In this case the thickness is defined through the section property definition. Conventional shell elements have displacement and rotational degrees of freedom.

Element type S4 is a fully integrated, general-purpose, finite-membrane-strain shell element. The element has four integration points per element, see following figure:

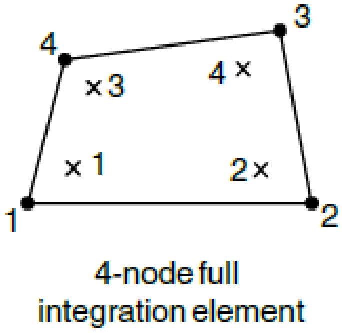

Figure 8: Shell element S4 (points indicate nodes, crosses indicate integration points)

A constant number of finite elements were used for all the cases:

- 12 or 16 elements on the width of the flange
- 16 elements on the height of the web
- 100 elements in the beam/column length/height

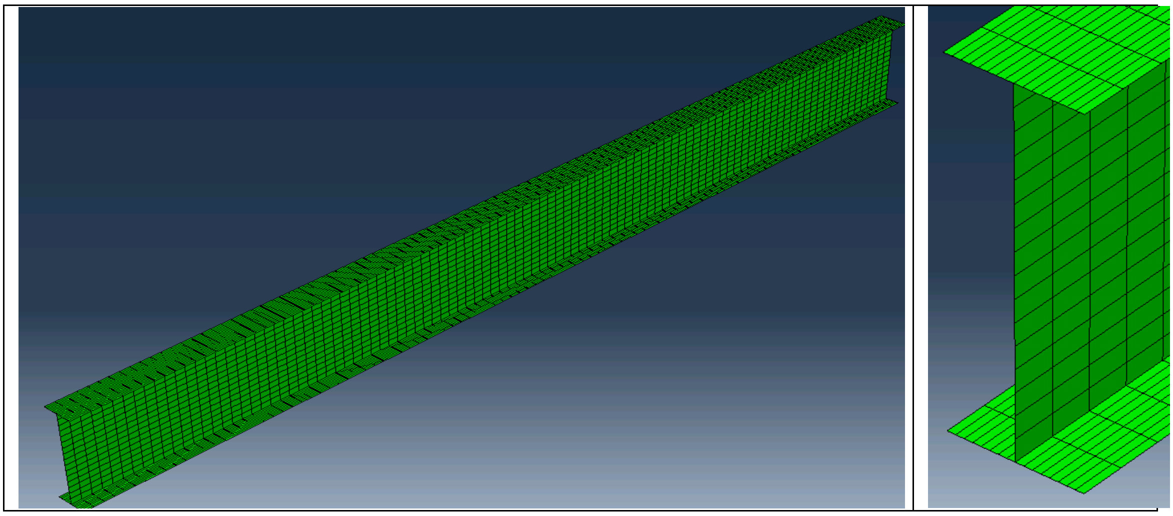

Figure 9: Illustration of mesh

2.2.2 Global and local imperfections

Geometrical imperfections were introduced in the model by modifying the nodal coordinates. The shape for the geometrical imperfections was considered as the first Eigen-mode of a linear buckling analysis and the amplitude of the imperfections has been considered as 80% of b/50 for plates corresponding to the flanges and 80% of b/100 for plates corresponding to the webs, following the recommendations of Part1-5 of the Eurocode to use 80% of the fabrication tolerances.

The global imperfection is also included in the model in case of it appears as the second or first Eigen-mode with amplitude of L/750 and reduced 70% if it is the second Eigen-mode.

For example in the following figures, it could be seen the results of linear perturbation analysis of HE1000AA hot-rolled section and 8 meters long in pure compression without lateral restraints.

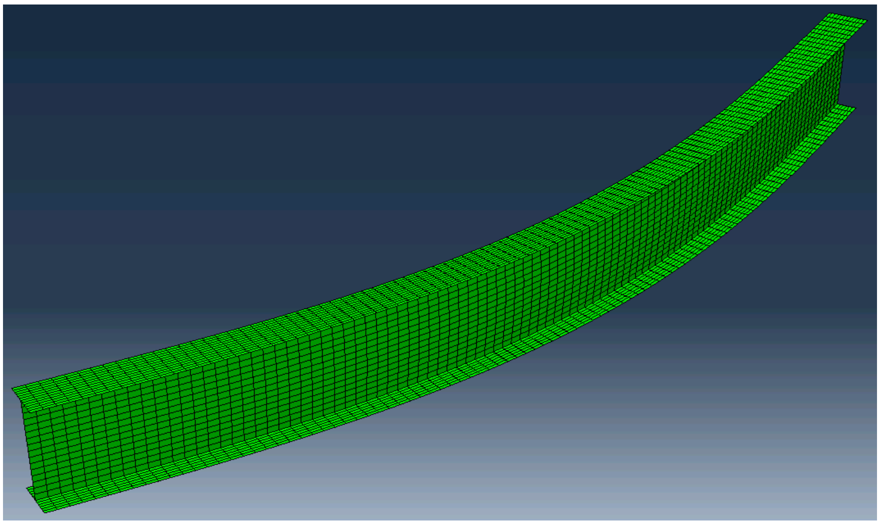

Figure 10: First Eigen-mode in perturbation analsysis to model the global imperfection shape

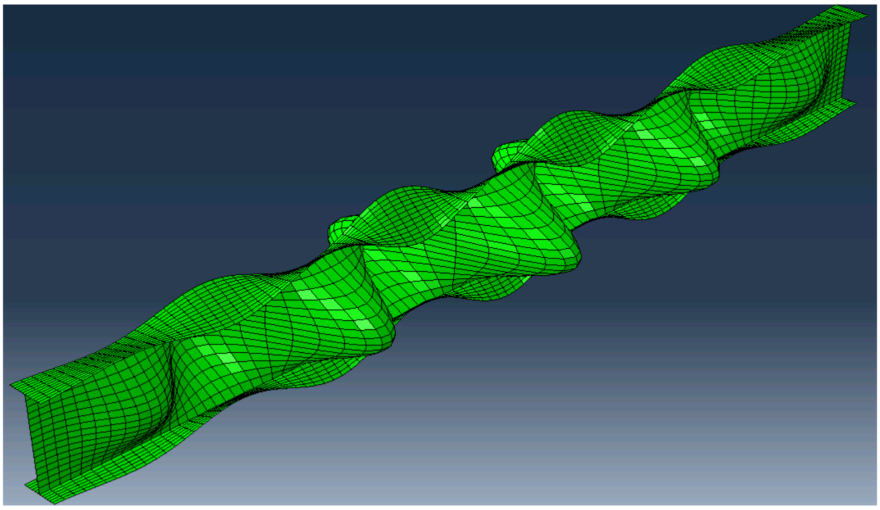

Figure 11: Second Eigen-mode in perturbation analysis to model the local imperfections shape

2.2.3 Material law

The material law was defined by elastic-plastic nonlinear stress-strain diagram, according to part 1-2 of the EC3, where enough data points were used for it and according to reduction coefficients with regard to steel temperature.

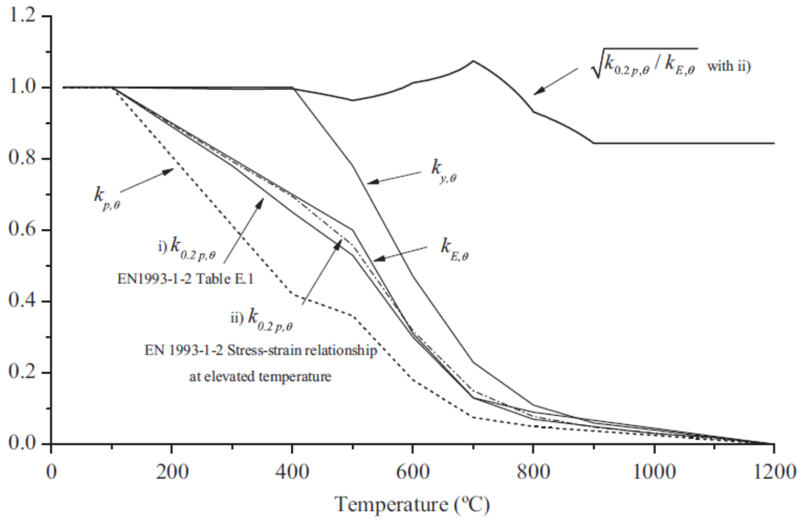

Figure 12: Evolution of carbon steel properties in function of temperature

Strain range	Stress σ	Tangent modulus
$\varepsilon \le \varepsilon_{p,\theta}$	$\varepsilon E_{a,\theta}$	$E_{a,\theta}$
$\varepsilon_{p,\theta} < \varepsilon < \varepsilon_{y,\theta}$	$f_{p,\theta} - c + (b/a)\left[a^2 - (\varepsilon_{y,\theta} - \varepsilon)^2\right]^{0,5}$	$\dfrac{b(\varepsilon_{y,\theta} - \varepsilon)}{a\left[a^2 - (\varepsilon_{y,\theta} - \varepsilon)^2\right]^{0,5}}$
$\varepsilon_{y,\theta} \le \varepsilon \le \varepsilon_{t,\theta}$	$f_{y,\theta}$	0
$\varepsilon_{t,\theta} < \varepsilon < \varepsilon_{u,\theta}$	$f_{y,\theta}\left[1 - (\varepsilon - \varepsilon_{t,\theta})/(\varepsilon_{u,\theta} - \varepsilon_{t,\theta})\right]$	-
$\varepsilon = \varepsilon_{u,\theta}$	0,00	-
Parameters	$\varepsilon_{p,\theta} = f_{p,\theta}/E_{a,\theta}$ $\quad \varepsilon_{y,\theta} = 0,02$	$\varepsilon_{t,\theta} = 0,15$ $\quad \varepsilon_{u,\theta} = 0,20$
Functions	$a^2 = (\varepsilon_{y,\theta} - \varepsilon_{p,\theta})(\varepsilon_{y,\theta} - \varepsilon_{p,\theta} + c/E_{a,\theta})$ $b^2 = c\,(\varepsilon_{y,\theta} - \varepsilon_{p,\theta})E_{a,\theta} + c^2$ $c = \dfrac{(f_{y,\theta} - f_{p,\theta})^2}{(\varepsilon_{y,\theta} - \varepsilon_{p,\theta})E_{a,\theta} - 2(f_{y,\theta} - f_{p,\theta})}$	

Figure 13: Parameters for carbon steel stress-strain curve at high temperature

2.2.4 Analysis steps

A sequence of three steps has been created in each model to consider the output of each step as the input of the next step. The defined three steps are the following ones for the case of introducing **variable thermal action** with the objective to obtain the resistance time:

- **EQUILIBRIUM STEP**: where the initial geometric imperfections, introduced as combination of Eigen-modes of a linear perturbation analysis, and the initial residual stress discretization get the equilibrium as initial situation before of any type of loading. This step is static.
- **PRE-LOADING STEP:** where the pre loading has introduced. This step is static.
- **THERMAL ACTION:** where the variable temperature field has introduced. This step is dynamic with aim of obtain the large deformations, reducing the time step value.

In case of **uniform heating** and wanting to get the ultimate resistance load, the sequence of three steps will be the following one:

- **EQUILIBRIUM STEP:** where the initial geometric imperfections, introduced as combination of Eigen-modes of a linear perturbation analysis, and the initial residual stress discretization get the equilibrium as initial situation before of any type of loading. This step is static.
- **UNIFORM HEATING STEP**: where the temperature increase from initial condition of 20 ºC to the established value. This is a static step.
- **LOAD RESISTANCE STEP**: where the increasing load or displacement has introduced. This step is dynamic with aim of obtain the ultimate resistance load.

3 New carbon steel material law to define new beam-column finite element in numerical analysis to take account of class 4 cross-sections behaviour

In this chapter, the development of modification of carbon steel material law for using beam finite elements in numerical analysis is explained. This research has been carried out by Prof. J.M. Franssen from University of Liege, in the scope of FIDESC 4 RFCS research project and details are available in [1].

3.1 Introduction

As previously commented in the introduction of this guidance, the use of slender steel sections has increased in recent years because their excellent strength to weight ratio. And the major issue with slender sections is local buckling that may occur in compression zones of the elements made of slender plates, in the flange under compression for elements in bending, in both flanges and also in the web for elements in compression.

That is why, to take local instabilities into account in a precise manner, the designer is left with no other choice than to use shell finite elements that can represent the local buckling phenomena as it is explained in the previous chapter. These elements are yet very expensive already for modelling single construction members, let alone for modelling complete structures. It is thus desirable to use cheaper beam elements modified to take local buckling into account.

The approach that is most often used is based on the concept of effective width; the width of the plates is reduced in such a way that the plastic capacity of the reduced section is equal to the capacity of the slender plate which exhibits local buckling. This approach has first been proposed and has been used for analytical analyses. Yet, because the effective width depends on the stress level which in turn depends on the effective width, this procedure is iterative, which is already a serious complication when it comes to analysing the situation of a single member under a defined loading.

If this approach has to be applied in beam finite elements used in transient or step-by-step analyses of complete structures, the additional level of iteration on the effective width leads to a severe modification, not only in the formulation of the finite element, but also in the formulation of the solution strategies of the code. Moreover, convergence problems may occur because it is not possible to derive the real tangent stiffness matrix of the elements.

3.2 New proposed carbon steel material law

It is here proposed to take local instabilities into account in beam type elements by means of an effective constitutive law of steel. The effective law has to be derived with the same objective as the effective width: the plastic capacity obtained with the effective law in the full section is equal to the capacity of the slender plate with the real material under local buckling. The following figure shows the two cited approaches to obtain the same objective:

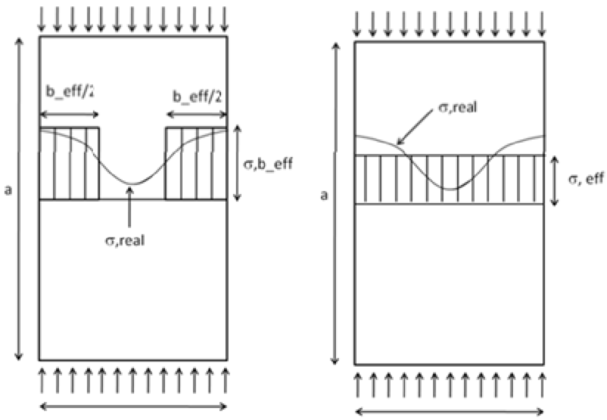

Figure 14: Effective width method (left) and proposed effective stress method (right)

Local buckling occurs only for compressive plates. As a consequence, the stress-strain relationship needs to be modified only in compression and remains unchanged in tension. This leads to a non-symmetrical law with respect to compression-tension.

The tangent modulus at the origin of the law is not modified (which comes from the fact that low compression stresses do not produce local instabilities), but the development of local instabilities is reflected by a reduction of the limit of proportionality, of the effective yield strength and of the characteristic strain corresponding to the relationship beginning of the horizontal plateau in the stress-strain.

The effective stress-strain relationship in compression depends on the slenderness and on the boundary conditions of the plates, either supported on four sides (as in a web) or supported on three sides (as in half flanges), and possibly also on the steel grade, but these conditions are known at the time of creating the model and can easily be entered by the user as new material properties. The material law also depends on the temperature, but this is already the case for the real law considered up to now and this can be easily accommodated by the numerical code.

The method used in this research to determine the effective stress-strain relationship is based on the simulation of isolated plates modelled in SAFIR computer code with shell elements, simply supported on three or four sides and subjected to progressive imposed shortening in one direction. The simulations are performed first at ambient temperature and then at various elevated temperatures. From each simulation of a plate, the effective strain at any time is considered as the shortening of the plate divided by initial length of the plate, whereas the effective stress is considered as the reaction force applied on the edge of the plate divided by the sectional area of the plate:

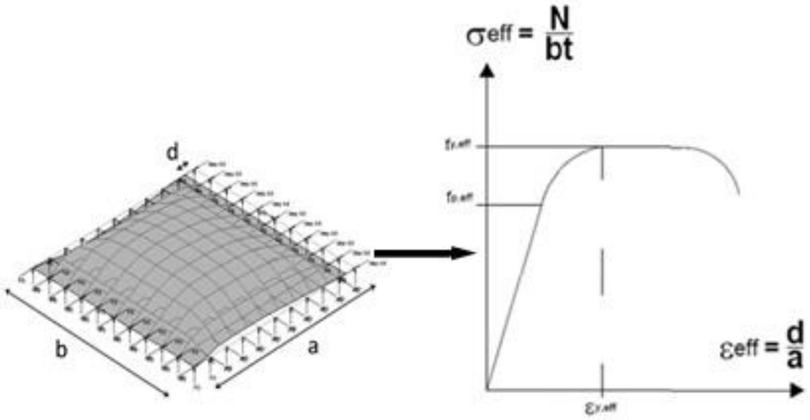

Figure 15: Illustration of the applied method to get the new material laws

If the obtained curves would be very different in shape from these currently used for the virgin material, new effective stress-strain relationship should be developed. It has been decided here to keep the relationship proposed by the Eurocode.

From the effective stress-effective strain curve obtained each plate, the effective yield strength, the effective proportionality limit and the effective strain corresponding to the beginning of the plateau were determined, depending on the relevant conditions of the plate. Additional illustrations and explanations are exposed in the following figures:

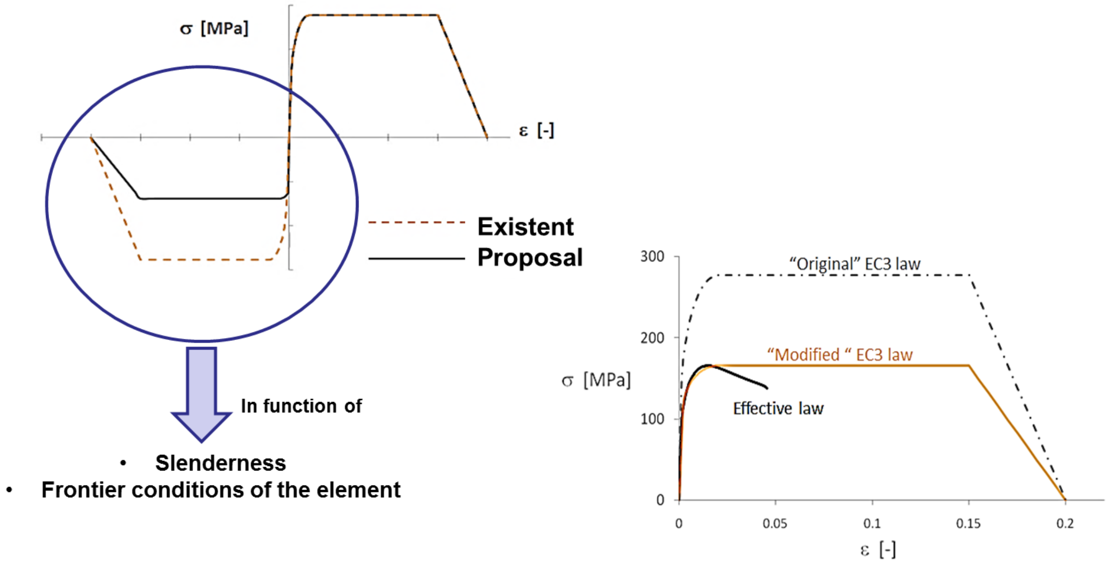

Figure 16: Differences between the material law

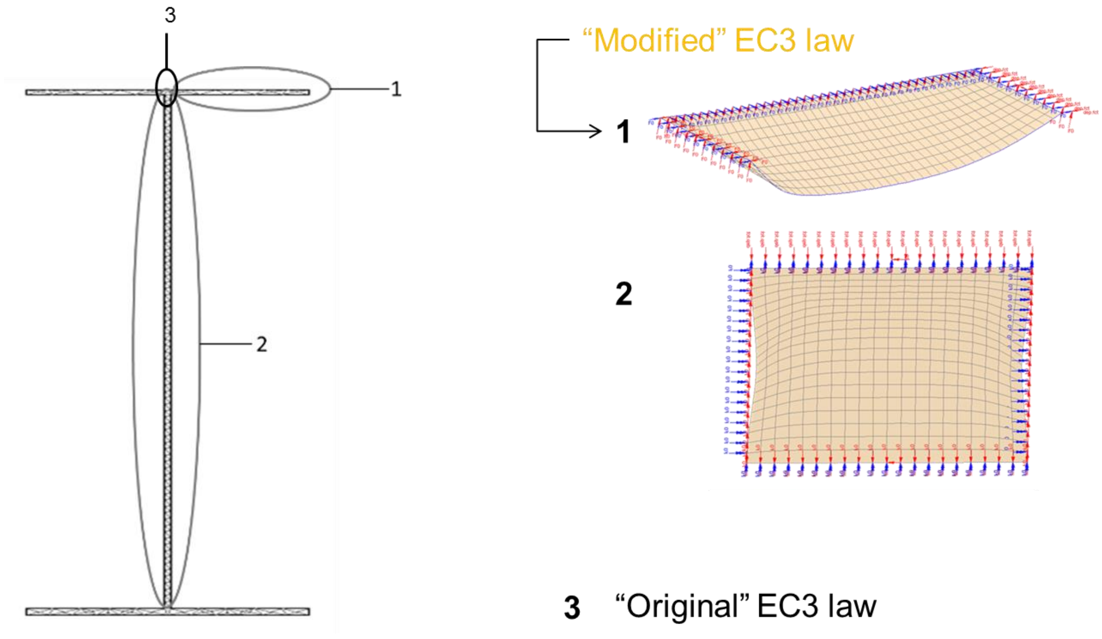

Figure 17: Illustration of buckling with "modified" EC3 law

The tables that give the values of the parameters of the effective law (limit of proportionality, effective yield strength and characteristic strains) at various values of the temperature and slenderness are established for both boundary conditions.

It has to be noticed that a simple adaptation of the subroutine at the material level can be made and easily introduced in any computer code. The user only has to introduce a different material model for the web and for the flanges, to give the slenderness of each plate as a new material property, and the software automatically takes care of the temperature, of the stress level and of the direction of the stress, tension or compression in each integration point. This procedure can be used also for analyses of structures at room temperature. It has to be underlined that, compared to existing methods, there is no stepwise variation of the behaviour at the interface between the four classes; in fact, there is no need to define the class because the adaptation of the material model is a continuous function of the slenderness.

The limit of this approach is that it cannot capture local buckling produced by shear forces, but this is also the case for the effective width approach.

3.3 How to define the new material law

The proposed effective law of steel is presented in this section. The first part specifies the approach used to take into account the reduction of the yield strength, the proportionality limit and the characteristic deformation in the compression zone of the stress-strain diagram. The second part describes the full stress- strain relationship in case of unloading.

3.3.1 Slenderness reduction factor

The influence of the slenderness on the yield strength for different temperatures for single plates simply supported on four sides submitted to compression in one direction is represented in the following graphic with steps of 100 °C. A steel grade of 355 MPa was used for these numerical simulations. The yield strength decreases with an increasing slenderness.

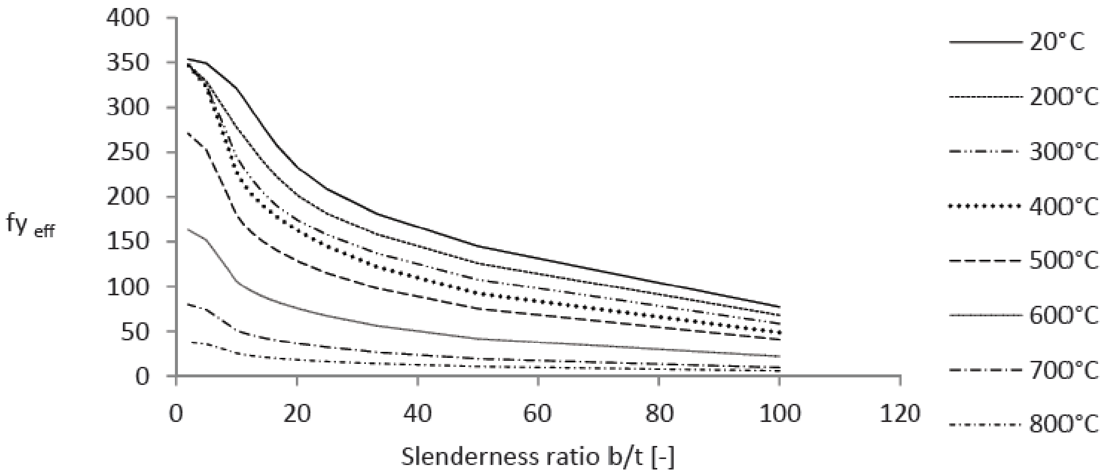

Figure 18: Yield strenght in function of slenderness ratio for different temperatures

The same results show the next graphic, using non-dimensional parameters for the slenderness as given in the following equations:

$$\lambda_E(\theta) = \pi \sqrt{\frac{E(\theta)}{f_y(\theta)}}$$

$$\bar{\lambda}_\theta = \frac{b/_t}{\lambda_E(\theta)}$$

$$k_{sl} = \frac{f_y k_{y,\theta}}{f_{y,eff}}$$

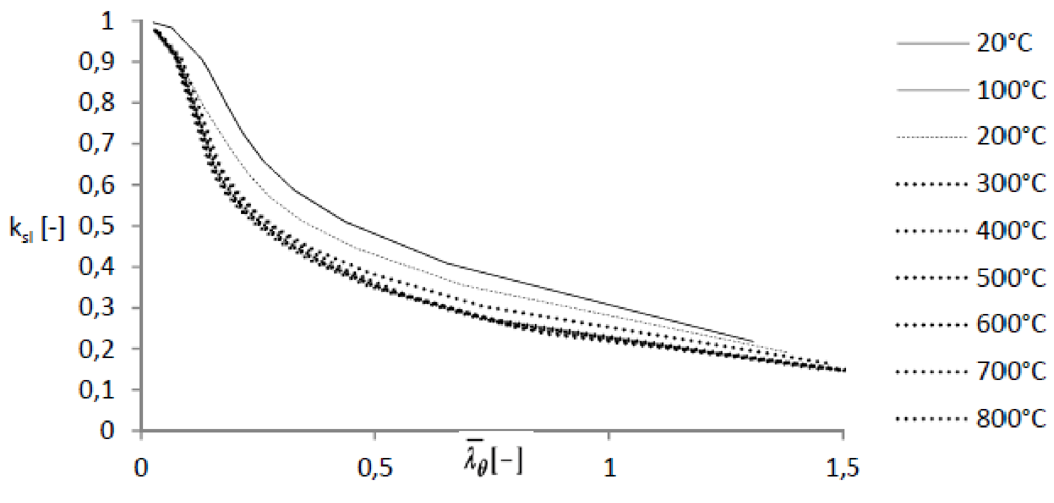

Figure 19: Evolution of k_{sl} in function of slenderness for different temperatures

It is observed that these curves can be separated into three different groups, one for ambient temperature (20°C ≤ θ ≤ 100°C), another one at 200°C and the last one for higher temperatures (300°C ≤ θ). It can also be seen that these curves look similar to the buckling curves of Eurocode 3 part 1.

It was thus chosen to approximate those results with the Perry-Robertson equation as it is the case for the buckling curves equation in the Eurocode:

$$\varphi = \alpha\left(1 + \beta\left(\bar{\lambda}_\theta + \gamma\right) + \bar{\lambda}^2{}_\theta\right)$$

$$k_{sl} = \frac{1}{\left(\varphi + \sqrt{\varphi^2 - \bar{\lambda}^2{}_\theta}\right)} \leq 1$$

At this step of the study, different values of the parameters α, β and γ are considered for ambient temperature, at 200 °C and at elevated temperatures (≥ 300 °C). The values differ also for the two considered support conditions. Between these different temperatures, a linear interpolation is used. Even if this model is developed for structures in the fire situation, it is crucial to know the values of the parameters at ambient temperature. Indeed, in fire situations, the structure is generally heated after be ng loaded a t ambient temperature. Also, some parts of the structure may be unaffected by the fire. The proposed effective law must thus be able to catch the behaviour of slender steel sections at room temperature.

Supports conditions	Temperature	α	β	γ
Flange (3 sides)	Cold (≤100ºC)	0.31	3.9	0.09
	200ºC	0.24	6.0	0.15
	Hot (≥300ºC)	0.19	10.0	0.14
Web (4 sides)	Cold (≤100ºC)	0.10	8.9	0.15
	200ºC	0.10	8.7	0.25
	Hot (≥300ºC)	0.07	16.5	0.21

Table 2: Parameters used to define the slenderness reduction factor

The same reduction factor k_{sl} was applied on the yield strength, the proportionality limit and the strain corresponding to the beginning of the plateau. The proposed stress-strain relationship in compression is shown at next figure for a plate simply supported on three sides with a slenderness ratio b/t of 20 and a temperature of 600 °C:

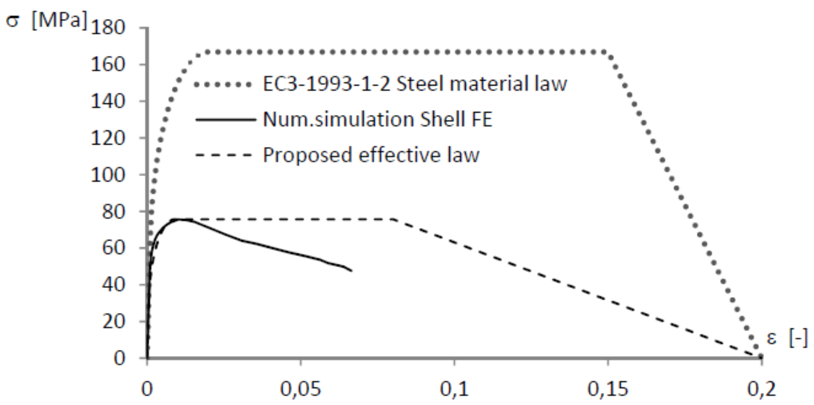

Figure 20: Example of stress-strain relationship with new effective law

It has to been noticed that at this step of the study, the proposed effective law reproduces the behaviour observed for single plates. Slender cross-sections are considered as an assembly of plates that are simply supported on three or four sides. The contribution of the web to increase the stiffness of the half-flanges is thus not taken into account. In reality, the plate is not simply supported on the side shared with another plate. The actual support condition is between a simple support and a fixed support. The assumption made in this study is thus conservative. This will be questioned in further studies if the proposed model appears to be too conservative.

3.3.2 Unloading after loading

The next Figure shows the behaviour observed for a single plate simply supported on three sides with a slenderness ratio b/t = 20 at a temperature of 500°C when the plate is at first submitted to tension and then submitted to compression:

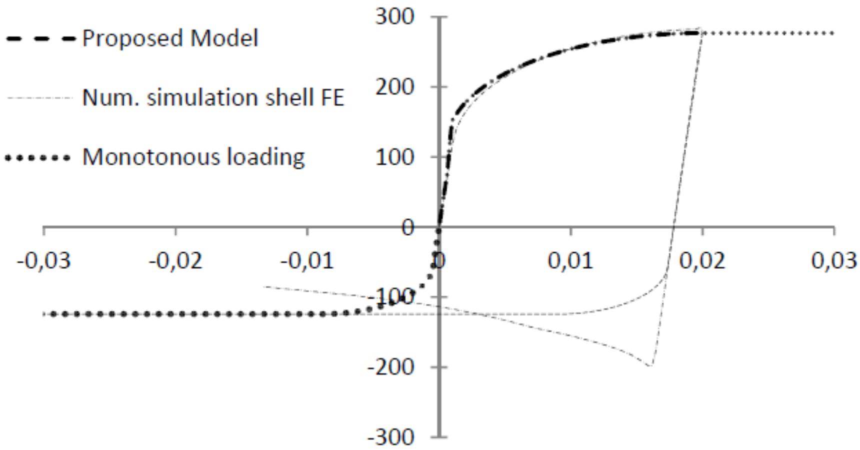

Figure 21: New material for tension then compression loadings

The difference between the behaviour observed and the proposed model is in the peak observed in compression after elastic unloading. The numerical result is obtained from the modelling of a single plate forced to enter into severe strain reversal. Also, the proposed model has been developed for structures in fire situation and not for cyclic loading. From a large number of numerical tests performed on structures or building assemblies subjected to fire, it was never possible to find any point of integration reaching this part of the diagram before collapse of the structure. Thus, this peak of the stress-strain diagram has not been taken into account and a simplified model was adopted. Once the stress becomes negative, thus when compression stress appears, the point follows the initial curve with an offset corresponding to the plastic deformation in tension.

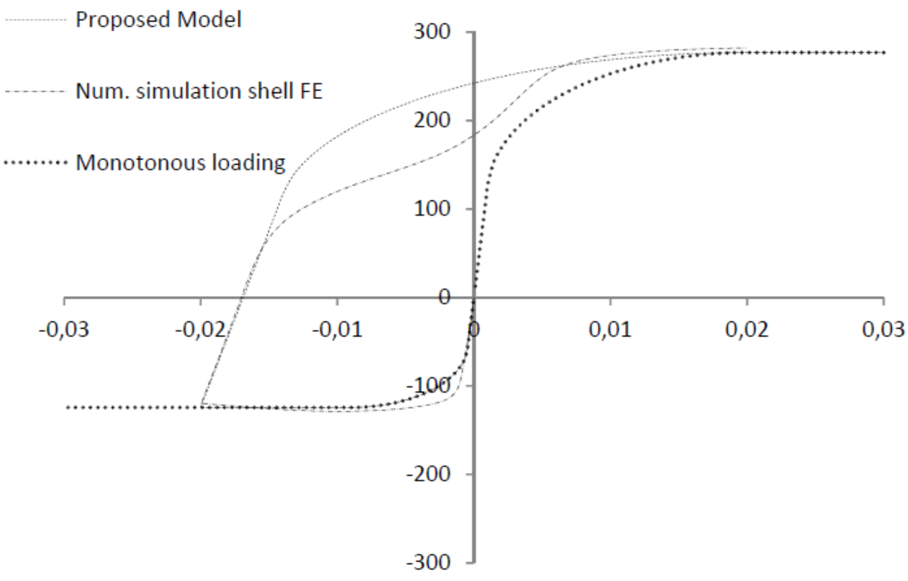

Figure 22: New material law for compression then tensiojn loadings

It has been observed that the reduction of stiffness is linked to the plastic deformation, the slenderness and the support condition but doesn't depend on the temperature. For the same plate under various temperature conditions, the ratio between original elastic stiffness (EN1993-1-2) at this temperature and the observed reduced stiffness during unloading remains constant.

Steel generally does not exhibit damage mechanisms linked to micro-cracks as it could be the case, for example, in concrete. In this case, the loss of stiffness in unloading is probably due to the plastic deformation of the plate that develops under first loading in compression. Nonetheless, as for concrete models, it was chosen from phenomenological observations to adopt a damage scalar to capture this effect.

Following equations give the damage scalar for plates and the Table 3 gives the values of the parameters to be used in these equations:

$$E_{unloading} = E_0(1 - D)$$

$$D = \frac{a\varepsilon_{pl}}{\varepsilon_{pl+b}}$$

$$b = c\bar{\lambda}_{\theta}^{d} + e$$

Support conditions	a	c	d	e
Web (4 sides)	0.84	0.0003	-3.5	0.0015
Flanges (3 sides)	0.95	0.0010	-1.9	0.0010

Table 3: Damage factor according to different support conditions

4 Worked examples

In this chapter, different examples of structural single members and portal frames are going to be analysed with previously defined new material law and compared them against shell finite element simulations.

4.1 Single members analysis

For all single members' analysis made with SAFIR Computer Code, there are some common features. They are the steel grade (S355), the temperature of 450 ºC uniform in all the length of the element, the global geometric imperfection defined as follows and initial residual stress:

$$y(x) = 0.8 \times \frac{L}{750} \times sin\left(\frac{\pi x}{L}\right)$$

4.1.1 Steel members with class 4 cross-sections under simple bending

The list of the investigated cases is given in the following table:

			Mult (kNm)				
Case	Section	Length [mm]	a)Shell	b) Beam Material Law EN1993-1-2	c) Beam Material Law Franssen et al.	b/a	c/a
1	1000x14+300x22	10000	2496.46	2936.88767	2089.21313	1.18	0.84
2	1000x12+300x18	10000	1843.62	2445.8546	1607.80541	1.33	0.87
3	1000x12+300x14	10000	1422.02	2074.3227	1275.52664	1.46	0.90
4	1000x8+300x18	10000	1608.47	2153.66997	1370.38713	1.34	0.85
5	1000x6+300x13	10000	943.87	1566.21378	859.62048	1.66	0.91
6	450x6+150x11	5000	251.52	318.49587	229.36947	1.27	0.91
7	450x6+150x9	5000	195.33	277.05439	188.52597	1.42	0.97
8	450x5+150x8	5000	157.57	239.95211	154.20289	1.52	0.98
9	450x4+150x6	5000	106	183.28417	103.25258	1.73	0.97
10	450x4+150x5	5000	89.33	162.31116	85.15998	1.82	0.95

Table 4: Investigated cases for pure bending

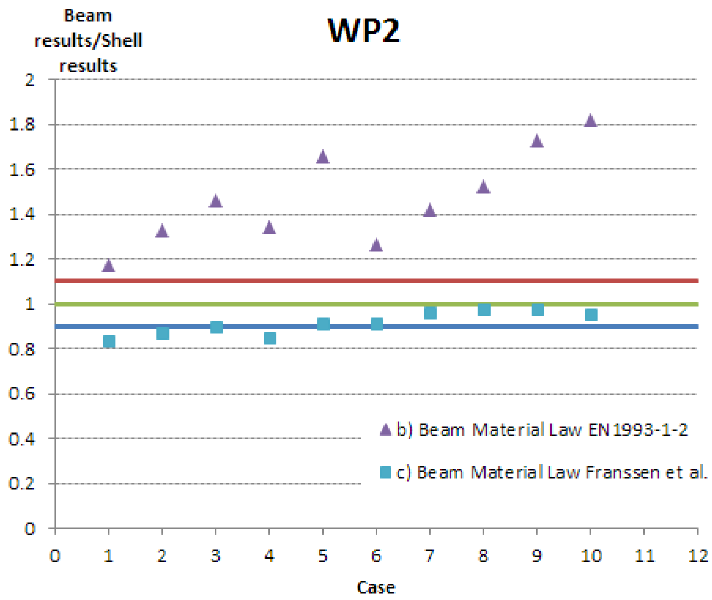

Figure 23: Comparisons between shell and new beam element models for pure bending

4.1.2 Steel members with class 4 cross-sections subjected to lateral torsional buckling behaviour

The list of the investigated cases is given in the following table:

				Mult (kNm)				
Case	Section	Length [mm]	psi	a)Shell	b) Beam Material Law EN1993-1-2	c) Beam Material Law Franssen et al.	b/a	c/a
2	610-450x5+150x5	5000	1	45.87	37.20111	27.49632	0.81	0.60
3	610-450x5+150x5	5000	0	85.36	63.65775	46.50845	0.75	0.54
4	610-450x5+150x5	5000	-1	97.9	92.79698	68.60357	0.95	0.70
5	450x5+250x5	8000	1	52.2	51.84236	37.05097	0.99	0.71
6	450x5+250x5	11000	0	73.29	66.64776	48.51575	0.91	0.66
7	450x5+250x5	13000	-1	80.12	82.94716	59.4503	1.04	0.74
8	1000x7+300x12	8000	1	381.42	378.94805	225.00396	0.99	0.59
9	1000x7+300x12	10000	0	510.45	537.8225	311.34366	1.05	0.61
10	1000x7+300x12	12500	-1	475.51	595.87827	383.49561	1.25	0.81

Table 5: Investigated cases for lateral torsional buckling

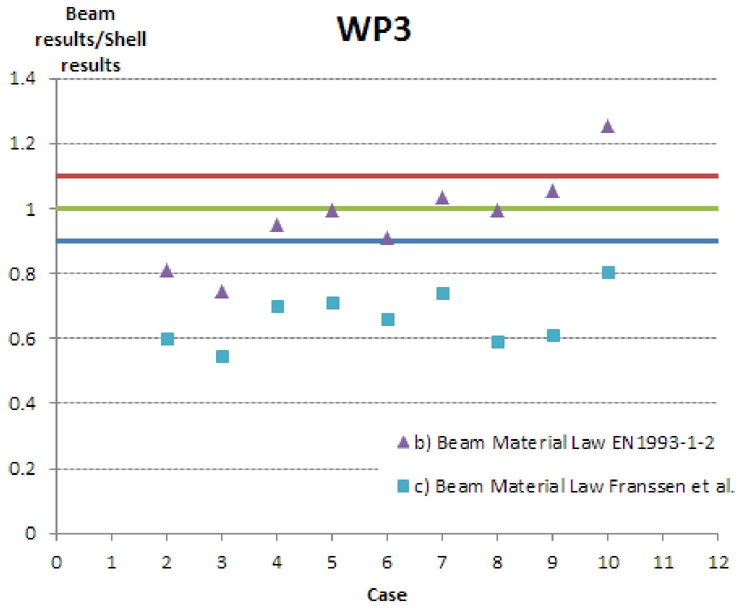

Figure 24: Comparisons between shell and new beam element models for lateral torsional buckling

4.1.3 Steel members with class 4 cross-sections under axial compression

The list of the investigated cases is given in the following table:

			Nult (kN)				
Case	Section	Length [mm]	a)Shell	b) Beam Material Law EN1993-1-2	c) Beam Material Law Franssen et al.	b/a	c/a
1	500x6+250x10	8000	441.61	419.92396	282.27317	0.95	0.64
2	500x4+250x6	6000	328.86	377.74801	191.35107	1.15	0.58
3	500x4+250x6	4000	421.26	575.12664	313.82214	1.37	0.74
5	500x4+250x12	6000	677.78	563.0695	392.94589	0.83	0.58
6	500x4+250x12	4000	940.59	945.53058	673.88441	1.01	0.72
8	500x10+250x6	6000	405.91	460.07	381.56	1.13	0.94
9	500x10+250x6	4000	627	775.56257	528.15594	1.24	0.84

Table 6: Investigated cases for axial compression

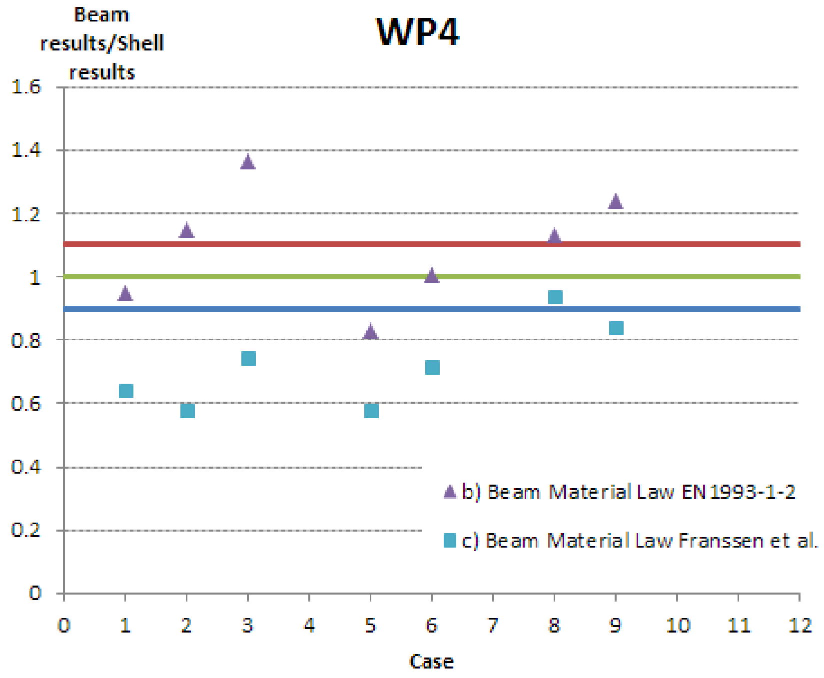

Figure 25: Comparisons between shell and new beam element models for axial compression

4.1.4 Steel members with class 4 cross-sections subjected to combined bending and compression

The list of the investigated cases is given in the following table:

						a) Shell		b) Beam Material Law EN1993-1-2		c) Beam Material Law Franssen et al.		b/a	c/a
Case	Section	Length [mm]	psi	LTB	beta	Nult (kN)	Mult (kNm)	Nult (kN)	Mult (kNm)	Nult (kN)	Mult (kNm)		
1	350x4+150x5	2700		allowed									
2	440-340x4+150x5	2700		allowed									
3	450x4+250x6	10000	1	restrained	0.6	192.66	62.35	348.89	113.04	178.94	57.98	1.81	0.93
4	450x4+250x6	10000	0	restrained	0.6	238.62	77.22	454.14	147.14	225.72	73.13	1.90	0.95
5	450x4+250x6	10000	1	allowed	0.6	92.8	30.03	125.97	40.81	71.32	23.11	1.36	0.77
6	450x4+250x6	10000	0	allowed	0.6	125.72	40.69	147.04	47.64	82.09	26.60	1.17	0.65
7	450x4+250x6	10000	1	allowed	0.4	169.43	22.94	158.66	22.85	84.67	12.19	0.94	0.50
8	1000x5+300x10	10000	1	restrained	0.6	486.47	349.98	1112.19	800.78	476.96	343.41	2.29	0.98
9	HE340AA *	10000	1	allowed	0.6	312.09	57.48	385.42	70.15	317.29	57.75	1.23	1.02
10	1000-750x5+300x10	10000	1	allowed	0.6	305.49	165.13	229.43	124.12	154.04	83.33	0.75	0.50

Table 7: Investigated cases for combined bending and compression

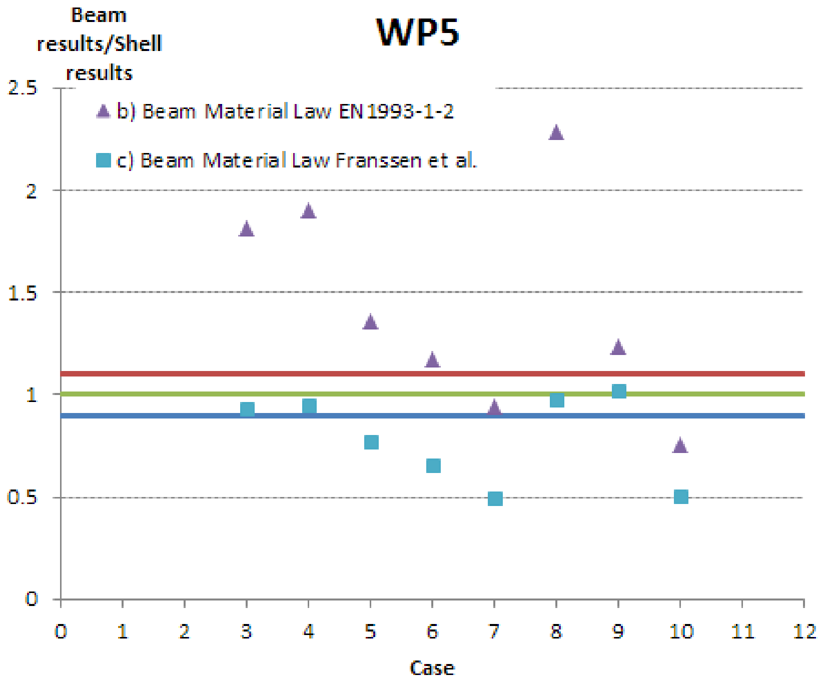

Figure 26: Comparisons between shell and new beam element models for combined compression and bending

Analysing the different results for all study cases, it could be said that it has demonstrated that EN1993-1-2 current material law is not suitable to use with beam finite element and class 4 cross section, because all the local instabilities could not be caught with them. That is why the coefficient b/a is bigger than one in the most cases. Although, it is known before the study started, it has been confirmed with results.

On the other hand, the validity of new material law has also been studied and it has been analysed the coefficient c/a, showing that it is smaller than unity, which is means that the resistance values using beam finite element plus new material law has more conservative results than the results obtained with shell finite element plus current EN 1993-1-2 material law.

However, in some cases the results obtained with this new approach is too conservative, being c/a smaller than 0.8. These conservative results are directly linked with assumptions taken in new material law formulation. As it is mentioned before, the boundary condition used in parametric plates analysis (web and flange boundary conditions), free rotation, is too conservative.

4.2 Global structural analysis

Two portal frames had been chosen as worked examples to analyse the behaviour of the simulation strategy and the use of the new material law for beam elements in a complex structure.

The objective of the following examples it is not the validation of the new beam element for the simulation of class 4 structures, developed in previous specific analysis with properly controlled simple member simulations, but the definition of a consistent simulation strategy for complex structures to obtain proper results.

In order to simplify the cases, neither residual stresses nor initial imperfections were considered in the analysis, to quantify only the effect of the geometrical discretization, the non-uniformity of temperatures and the new implemented material law for beam elements in the final results, in comparison with calibrated shell based finite element models.

The dead loads were considered uniformly distributed in the different elements, while live forces were considered applied simulating the effect of purlins in the structure:

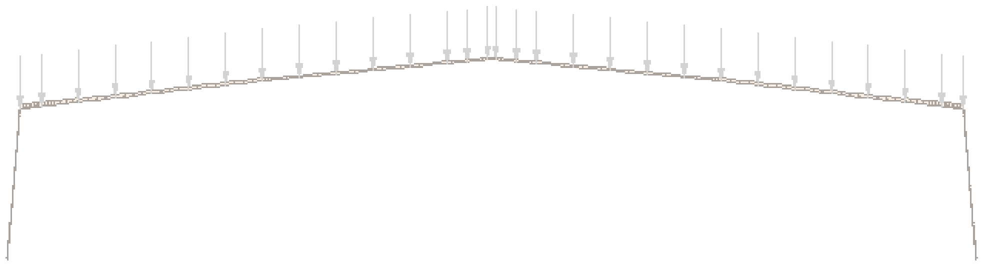

Figure 27: Load pattern definition in the portal frame examples

The same discretization and simulation strategy was developed in both cases, sometimes with close and no so close results, in order to obtain general conclusions about general rules to be implemented as guidance for future analyses. In order to have an accurate description of the structural behaviour of the frame, a validated complex shell based finite element models were developed to serve as validated reference.

4.2.1 Influence of the geometrical discretization

In order to simulate the case structure with the new beam element, a proper geometrical discretization may be developed. The variable section real geometry of the frame may be simplified as a succession of constant section beams to be properly modelled with simple beam FEM elements:

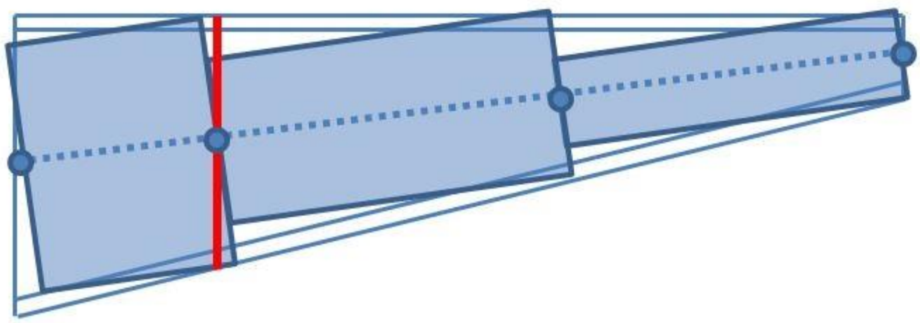

Figure 28: Geometrical discretization strategy

A line diagram is define from the real geometry configuration, linking the centre of mass of the edge sections or using an edge straight line as reference, while the definition of the sections must be consistent. As conservative criteria, the discretized structure must be limited to the actual geometry, so no added material will be presented in the discretized model definition.

In order to analyse the effect of the geometrical discretization in the accuracy of the models, a 39.6 meters one span single portal frame of S355 steel grade with high variable sections in columns and beams will be initially considered.

The geometrical configuration of the frame can be observed in the following pictures. More information about this work example can be obtained from the benchmark study developed in the FIDESC4 project (first deliverable).

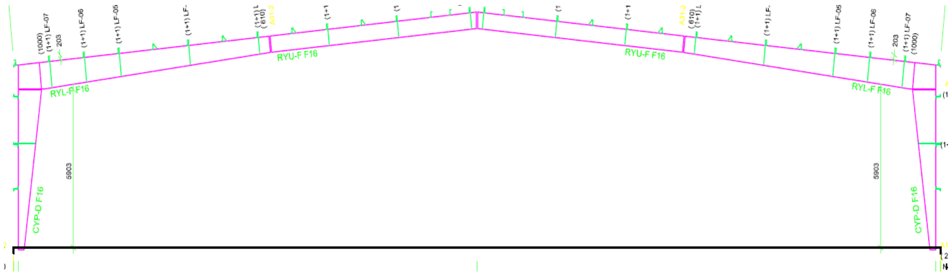

Figure 29: Geometrical definition of the one span frame example 1/2

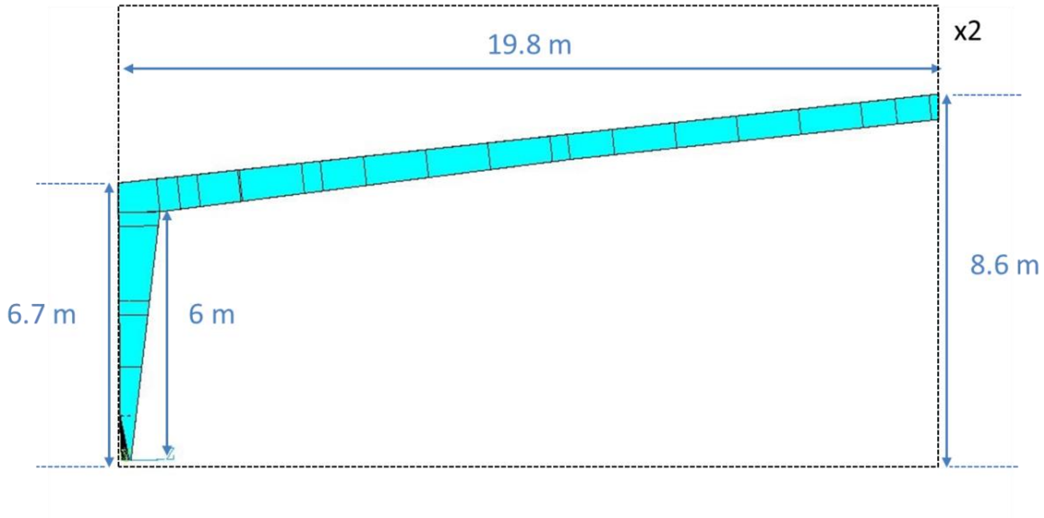

Figure 30: Geometrical definition of the one span frame example 2/2

A discretization based of 5 column geometrical sections (S1 to S5) and 6 beam geometrical ones (S6 to S11) resulted highly conservative to describe the real behaviour of the actual frame, with errors in the analysis of the failure load for cold conditions in order to 50% on the safe side. More detailed models could obtain closed results but compromising the simplicity of beam element based FEM analysis.

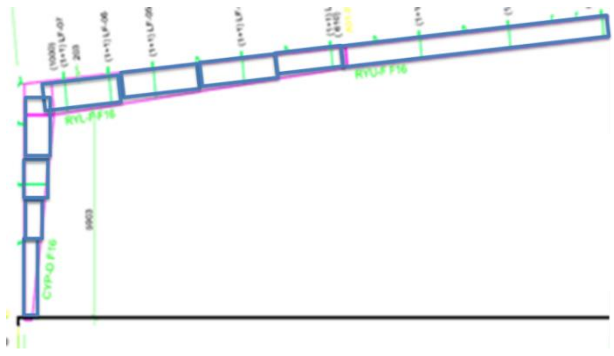

Figure 31: Geometrical discretization of the one span portal frane

A balanced ratio between case definition effort and results accuracy could be obtained implementing a more detailed geometrical discretization in the most loaded sections of the structure, where the lack of non-considered material has an important role in the global behaviour of the complete structure.

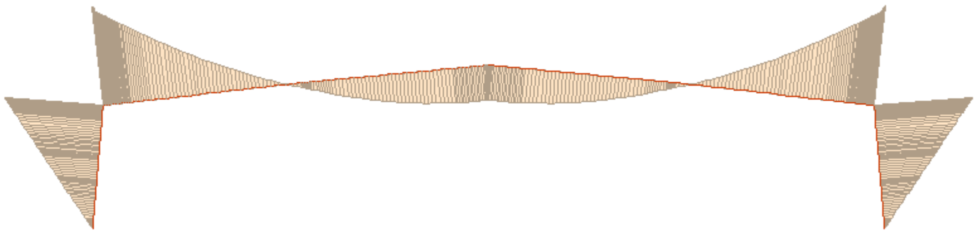

Figure 32: Bending moment focused points on the one span portal frame

In the case of a light variable section structure, close results could be obtained with the same strategy for the definition of the discretised model.

In this case, a S355 steel grade two span single portal frame with constant section columns and light variable section beams will be used as example. The specific geometrical definition of the case can be observed in the following pictures.

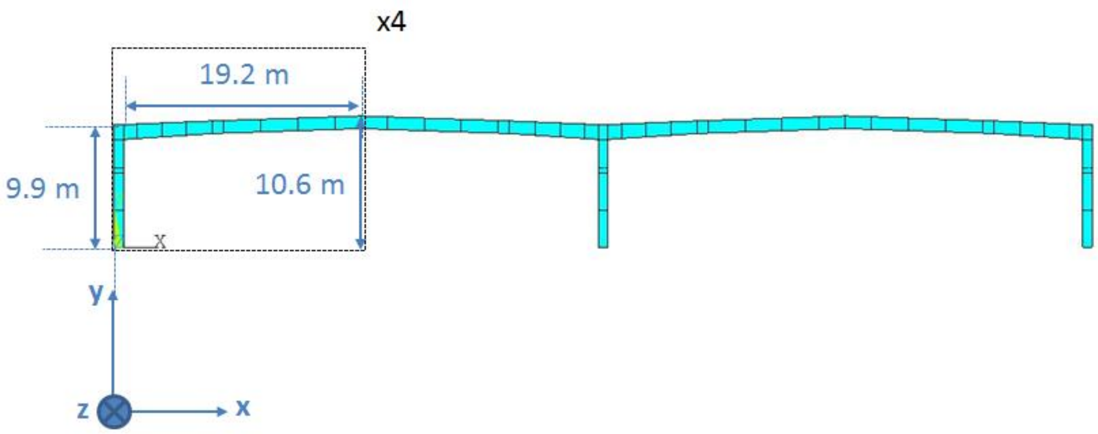

Figure 33: Geometrical definition of the two span portal frame 1/2

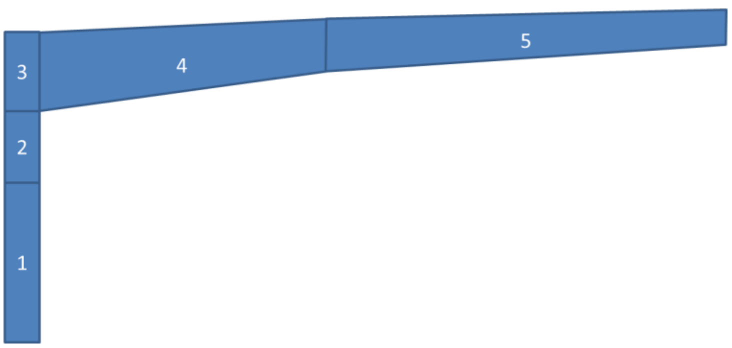

Figure 34: Plates defining the two span portal frame 2/2

Coordinates (m)	Plates			Plates			
	1	2	3	4		5	
X_{min} - X_{max}	0 – 0.775	0 – 0.775	0 – 0.775	0.775 – 7.975		7.975 – 19.795	
Y_{min} – Y_{max}	0 – 6.4	6.4 – 8.7	8.7 – 9.9	8.7 – 9.9	9.18 – 10.2	9.18 – 10.2	9.58 – 10.6
Z_{min} – Z_{max} (flange width)	-0.19 – 0.19 (0.38)	-0.19 – 0.19 (0.38)	-0.19 – 0.19 (0.38)	-0.125 – 0.125 (0.25)		-0.125 – 0.125 (0.25)	
Thicknesses	t_w = 0.008 t_f = 0.025	t_w = 0.02 t_f = 0.025	t_w = 0.02 t_f = 0.025	t_w = 0.008 t_f = 0.02		t_w = 0.006 t_f = 0.015	

Table 8: Details on dimensions of the two span portal frame

In this second example , a simpler discretization based of 1 column geometrical section (S1) and 4 beam geometrical ones (S2 to S5) resulted accurate enough to describe the real behaviour of the actual frame, with errors in the analysis of the failure load for cold conditions below 2% on the safe side.

So, the geometrical discretization of the actual structure it is an important point to be considered and it will be properly developed following general engineering principles, validating the general strategy for cold conditions prior to begin the analysis at high temperatures.

4.2.2 Influence of the heating conditions

The way in which the heat conditions are applied has a certain influence in the accuracy of results of the new beam element based simulations against shell based FEM validated analyses.

4.2.2.1 Uniform temperature distribution

In order to analyse the influence of a uniform temperature evolution in the new beam element based model, the failure load was calculated in the case of the two span portal frame example considering constant temperatures of 350 °C and 700 ºC in the whole structure.

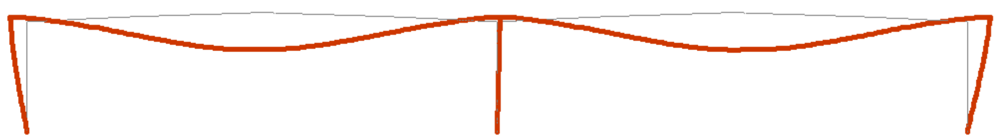

Figure 35: Deformed shape of the two span portal frame at 350 °C

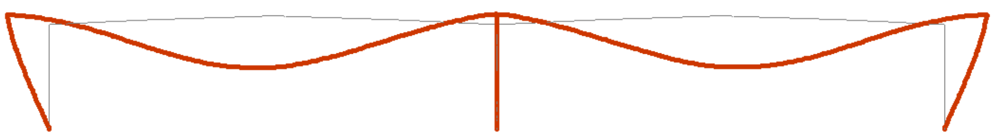

Figure 36: Deformed shape of the two span portal frame at 700 °C

In both cases, the accuracy of the new beam based model was below 13% on safe side against the shell element based validated finite element model.

4.2.2.2 *Non-uniform temperature distribution*

In order to analyse the influence of a non-uniform temperature evolution in the new beam element based model, the failure time was calculated in the case of the two span portal frame example considering two specific real fire scenarios (a fire at mid-span of a frame and a fire near a column).

The design fire is defined with a heat release rate of 750 kW/m², a fire area of 36 m² (diameter of about 6.77 m) and a flame height of 7 m.

The heat transfer to the structural sections is calculated as the maximum of different methods (Cfast, Hasemi and Heskestad) depending on the distance from fire (radiation and or hot layer).

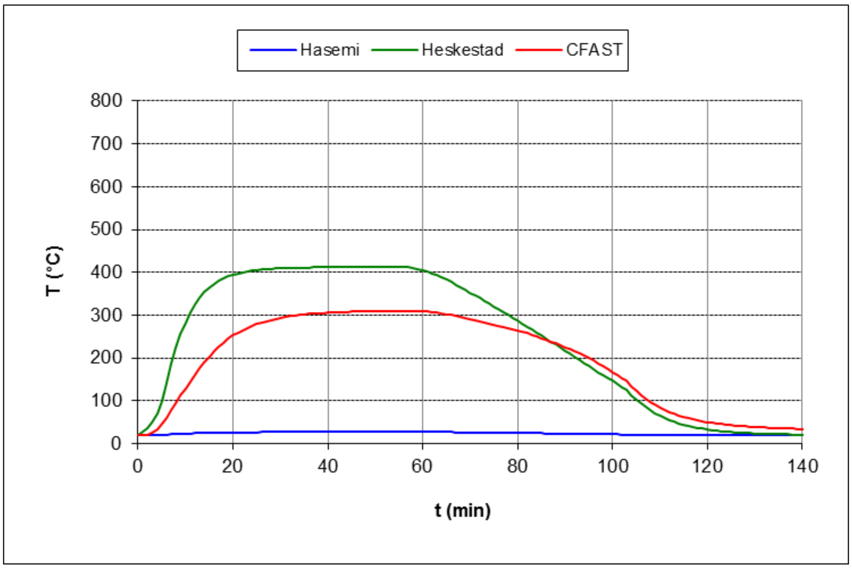

Figure 37: Evolution of temperature in function of time in the selected real fire scenario

The variable temperature distribution along the element length is considered as a linear interpolation between every two cross-sections, separated every 1 meter long, which their steel temperature values in function of time have been extracted previously from fire development analysis.

The temperatures are given each 60 seconds for a specified section in each beam/column. If lower time steps than 60 seconds are necessary a linear interpolation of the temperature is possible. Between two given temperatures in the sections a spatial linear interpolation is to be achieved.

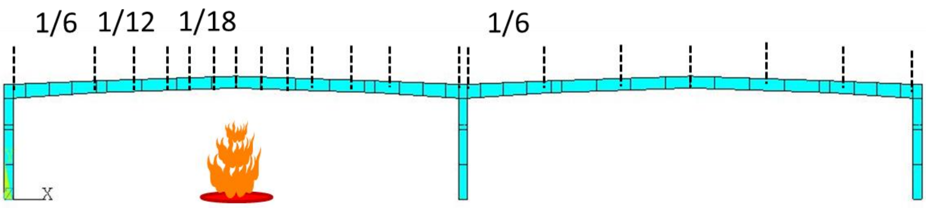

Figure 38: Reference sections for temperature definition – mid span fire

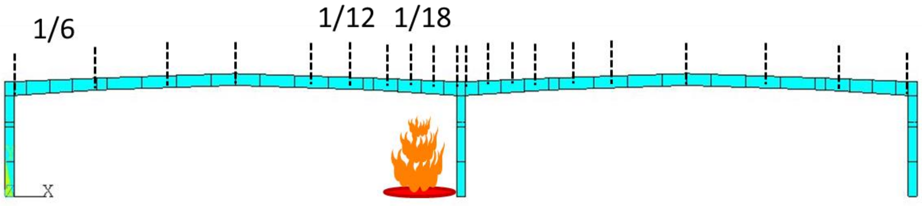

Figure 39: Reference sections for temperature definition – internal column fire

XT(i)	1	2	3	4	5	6	7	8	9
X (m)	0.775	7.175	13.575	19.975	26.375	29.575	32.775	34.908	37.042

10	11	12	13	14	15	16	17	18	19	20
39.175	39.95	42.08	44.216	44.99	48.19	51.39	57.79	64.19	70.59	76.99

Table 9: Reference sections coordinates for temperature definition

The deformed shape at failure is illustrated in the following figures for both shell and new beam finite element models:

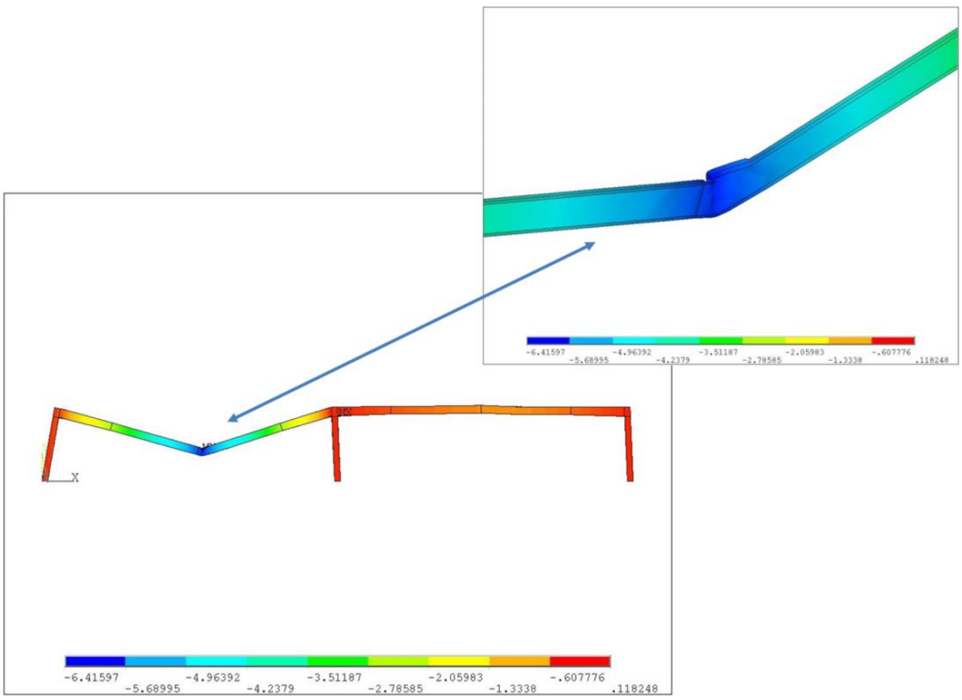

Figure 40: Deformed shape of the two span portal frame shell model (35 kN) – mid span fire

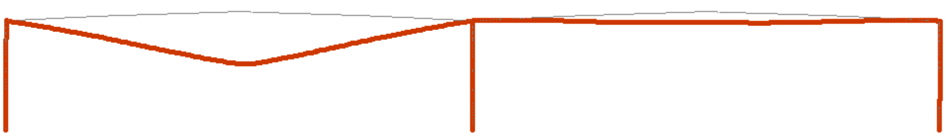

Figure 41: Deformed shape of the two span portal frame beam model (35 kN) – mid span fire (x5)

The following graph illustrates the vertical displacement in function of temperature at mid span for both models:

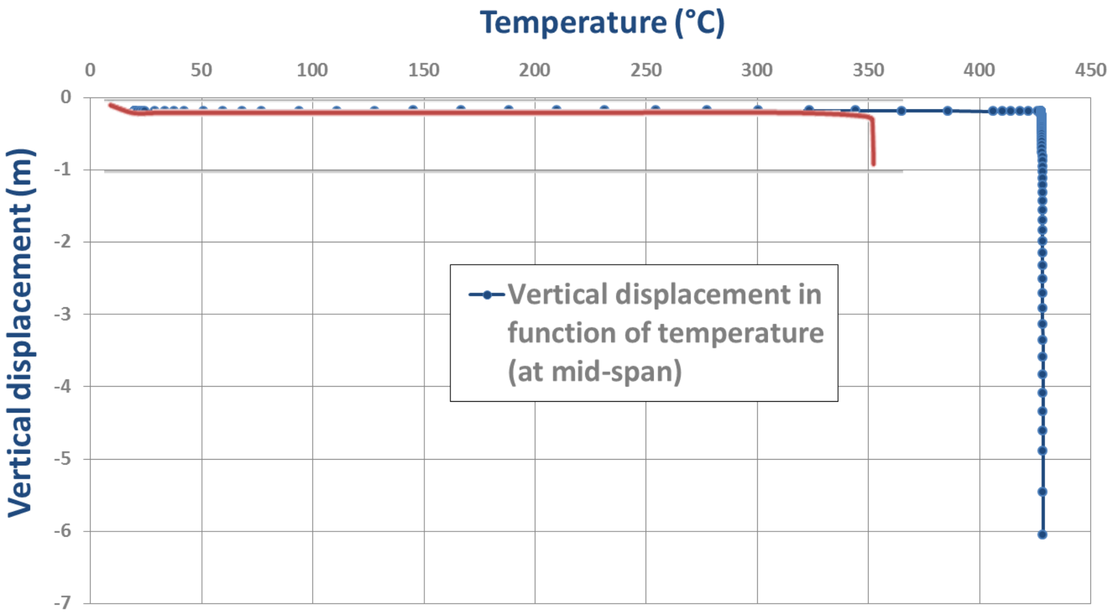

Figure 42: Vertical displacement (m) in function of temperature (°C) at mid span (red is beam model, blue is shell model)

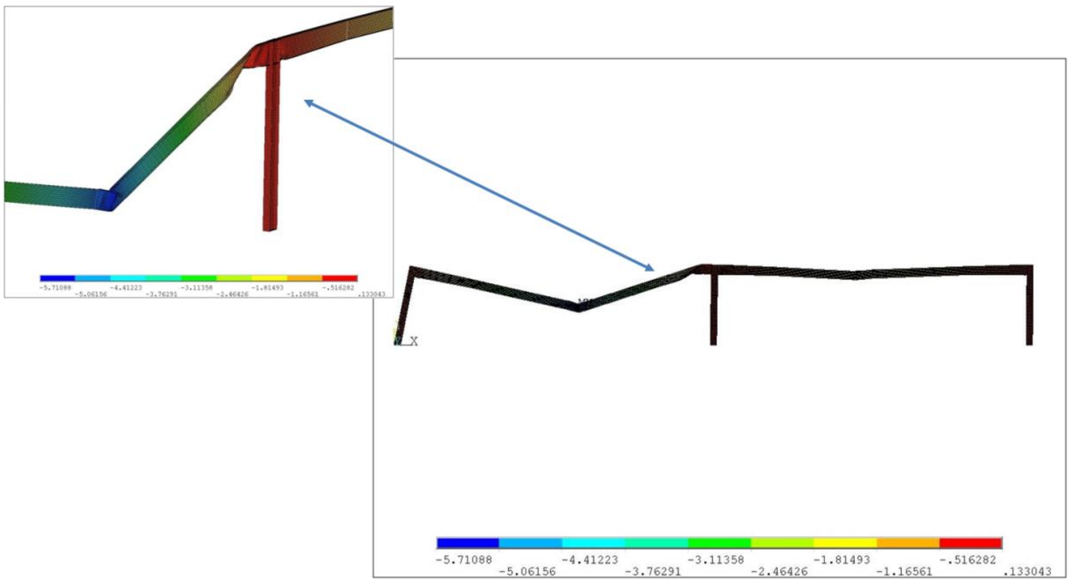

Figure 43: Deformed shape of the two span portal frame shell model (35 kN) – internal column fire

Figure 44: Deformed shape of the two span portal frame new beam model (35 kN) – internal column fire (x10)

In all the developed analysis, errors up to 60% in the calculation of failure times have been reported in the scope of this study due to the influence of non-uniform variation of temperature in beam sections, always on the safe side. This influence appears to be highly dependent of heating rate and the way the geometrical discretization is made to the implementation of variable temperatures.

5 Conclusions

This document has explained the general strategy of fire structural engineering design, taking into account the three basic stages (Fire Behaviour, Thermal Response and Structural Behaviour), explaining each of them and the different calculation tools; which are available for steel structures design; in function of complexity, from tabulated data or standard curves to finite elements method.

As conclusion, FIDESC4 “Fire Design of steel members with welded or hot-rolled class4 cross-section” Project partners have investigated the effects of uniform heating and real fires on different single elements and portal frames composed by class 4 steel cross sections, using shell finite elements, to take into account the local instabilities, which normally occurs in this kind of slender sections.

In addition, the definition and validity of the specific new material law for the carbon steel approach developed by University of Liege and included in SAFIR computer code was studied, taking into account the influence of local buckling of thin wall of steel members and explaining how to define it in function of slender of the plates and type of boundary conditions of the different plates of the section.

This constitutive material model has been applied in case of single steel member and portal frames under uniform heating and variable heating field and the obtained results are quite satisfactory.

Some comparisons have been made with numerical results obtained by much more expensive shell finite elements and with beam finite element plus new material law. The results are generally satisfactory although sometimes too conservative. A possible way of improvement could be to take into account the support provided by the plates to each other which create some supports that are not completely free in rotation; the slenderness of the plates may thus be lower than considered in these applications.

In case of complex structures, the geometrical discretization of the actual structure it is an important point to be considered and it will be properly developed following general engineering principles, validating the general strategy for cold conditions prior to begin the analysis at high temperatures.

On the other hand, the way in which the heat conditions are applied has a certain influence in the final accuracy of results of the new beam element based simulations against shell based FEM validated analyses. This influence appears to be highly dependent of heating rate and the way the geometrical discretization is made to the implementation of variable temperatures.

In consequence, more research is needed related to the optimisation of this carbon steel material law in case of uniform heating and, specifically with variable heating action, with the aim of using more cost-effective beam finite element without losing accuracy for this type of slender sections, class 4 steel cross sections.

6 References

[1] Franssen, J.-M.; Cowez, B. (2012). *"Consideration of local instabilities in beam finite elements by means of effective constitutive law"*, 7th International Conference on Structures in Fire, Zurich, Switzerland.

[2] *"Fire Design of Steel Structures"* ECCS Eurocode Design Manuals - J.M. Franssen and P. Vila Real-1st Edition – 2010.

[3] *"Guide to the advanced fire safety engineering of structures"- ISstructE – August 2007*

[4] *"Enclosure Fire Dynamics" B. Karlsson and J.G. Quintiere- CRC Press - 2000*

[5] *"Getting Started with Abaqus – Interactive Edition" Version 6.7. – Dassault Systèmes and Simulia*

[6] *"SFPE Engineering Guide to Performance –based fire protection" Second edition 2007 – Society of Fire Protection Engineers – NFPA & SFPE.*

[7] *"Fire Engineering Design Guide" Third Edition 2008 – New Zealand Centre for Advanced Engineering – University of Canterbury*

[8] *"Structural Fire Engineering*" – T.Lennon - BRE Fire and Security – 2011 - ICE publishing

[9] *"Designers´guide to EN1991-1-2, EN1992-1-2, EN1993-1-2 and EN1994-1-2"* – T.Lennon, D.B. Moore, Y.C. Wang and C.G. Bailey – Ed. Thomas Telford – 2007.

[10] *"CEN, EN1090-2: execution of steel structures and aluminium structures- part 2: technical requirements for steel structures".* European Committee for Standardisation, Brussels; 2008

[11] "Calculating the buckling strength of steel plates exposed to fire" – Quiel SE, Garlock MEM. Thin-walled struct 2010;48(9): 684-95.

[12] *ECCS,"Ultimate limit state calculation of sway frames with rigid joints" Publication nº33. European Convention for Constructional Steelwork Technical Committee Nº8, 1984.*

[13] *ECCS "Manual on stability of steel structures" Publication nº22 European Convention for Constructional Steelwork Technical Committee Nº8, 1976.*

[14] *ECCS "New lateral torsional buckling curves kLT-numerical simulations and design formulae" European Convention for Constructional Steelwork Technical Committee Nº8, 2000.*