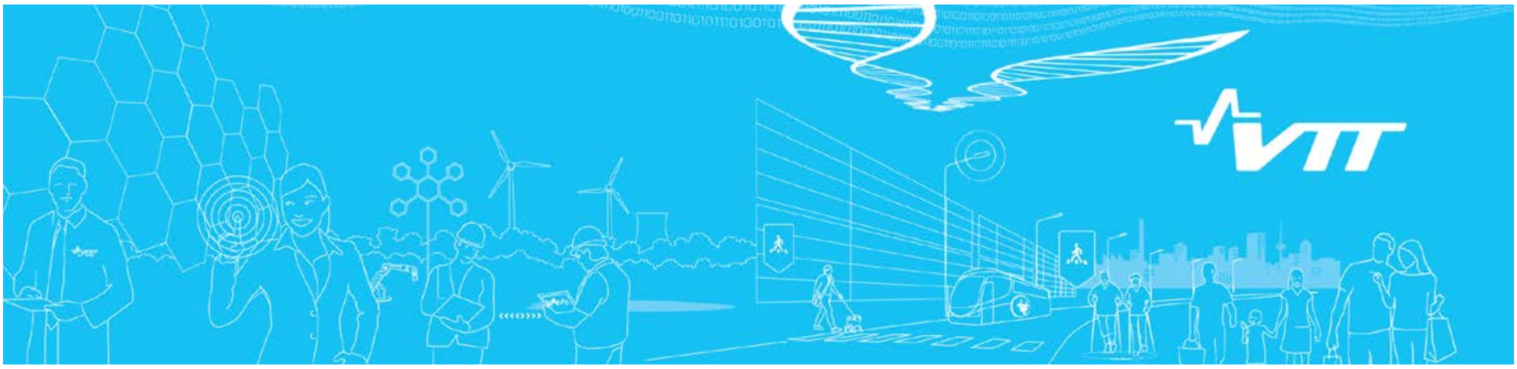
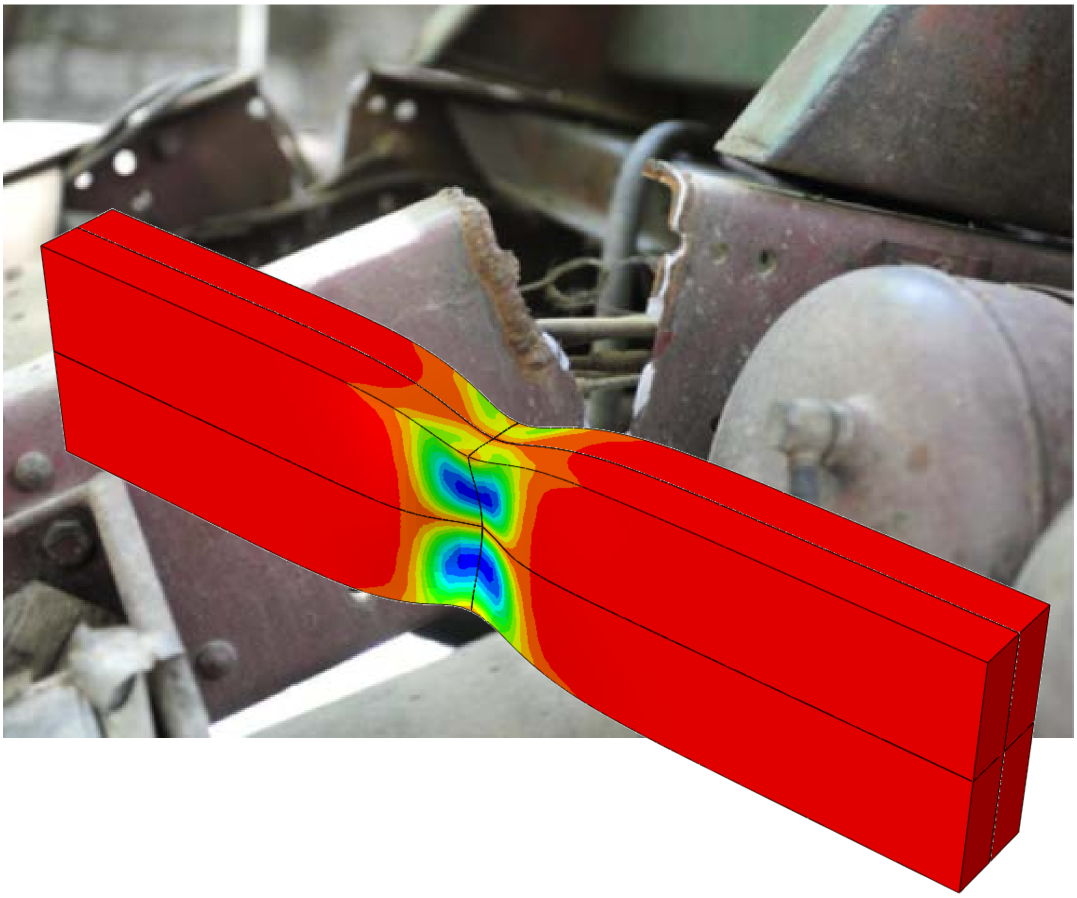


RESEARCH REPORT

VTT-R-01177-17



Experiences from numerical modelling of details with ductile failure

Authors: Petr Hradil, Ludovic Fülöp, Asko Talja, Juha Kurkela

Confidentiality: Public

Report's title	
Experiences from numerical modelling of details with ductile failure	
Customer, contact person, address	**Order reference**
Digital, Internet, Materials & Engineering Co-Creation (DIMECC)	BSA P2 SP2 Task 4
Project name	**Project number/Short name**
Virtual testing lab for novel materials and products	104679-1.2.3 / VILMA
Author(s)	**Pages**
Petr Hradil, Ludovic Fülöp, Asko Talja, Juha Kurkela	26
Keywords	**Report identification code**
steel, FEM, true stress-strain, ductility, necking	VTT-R-01177-17

Summary

The virtual testing methods of materials and structural details in VILMA project presented in this report were used to provide insight into the basic questions whether the Eurocode's material requirements are correct or not, could they be eased or not, and how much they could be eased. The methods are validated against real experiments.

The experiences from the conducted studies are summarized herewith for the purpose of numerical evaluation of material behaviour in details with high localized strains and possible diffuse necking.

Two basic numerical approaches presented in this report are: (a) the practical limitations for the numerical calculations to ensure that the ductile failure does not occur in the material and (b) the method to predict the minimum required material ductility in a certain situation.

This report can serve as a guidance for the finite element modelling of structural details that are subjected to tension, with the risk of initiation and development of diffuse necking.

Confidentiality	Public

Espoo 3.3.2017

Written by	Reviewed by	Accepted by
Petr Hradil	Heli Koukkari	Edgar Bohner
Senior Scientist	Principal Scientist	Research Team leader

VTT's contact address

P.O. Box 1000, FI-02044 VTT, Finland

Distribution (customer and VTT)

Online at DIMECC Research Portal

The use of the name of VTT Technical Research Centre of Finland Ltd in advertising or publishing of a part of this report is only permissible with written authorisation from VTT Technical Research Centre of Finland Ltd.

Preface

This report presents an approach to handle the ductility requirements of steel under tension, in particular in the necking areas of cross-sections. The presented calculation methods are based on the experiences from the numerical studies with regard to different design situations. They provide alternative and complementary solution to the Eurocode's material requirements that is practical especially for the building and product designers using finite element software.

The experiences from the conducted studies are summarized herewith for the purpose of numerical evaluation of material behaviour in details with high and localized strains and possible diffuse necking. The methods are validated against real experiments.

Two basic numerical methods presented in this report provide: (a) the practical limitations for the numerical calculations to ensure that the ductile failure does not occur in the material and (b) prediction the minimum required material ductility in a certain situation.

The report has been prepared in the project "Virtual testing lab for novel materials and products", called VILMA (2014–2016). The main research objective of VILMA was to develop a virtual testing platform for a fast and effective introduction of new structural steels and steel products to the market. The main practical objective was to use the platform to develop recommendations of the material ductility requirements in the Eurocodes for high strength steels.

The VILMA project belongs to the program "BSA - Breakthrough steels and applications and its project portfolio P2, "Design beyond present codes – enabling efficient utilisation of new materials". The industry-driven project portfolio was planned to rise to critical future needs of steel end-users such as product manufacturers, designers and building owners (market pull). The overall goal of BSA program is to enable a renewal of the Finnish metal and engineering industries through major improvements in their offerings and global competitiveness brought about by the intelligent use of novel advanced steel products. Key emphasis is on end-users in selected business areas: bioenergy, power generation, mining, lifting, handling and transport, offshore and marine, waste recycling, arctic technologies and processing industry (http://www.fimecc.com/programs/bsa). The BSA program is a part of the large national DIMECC innovation eco-system that represents a new type of public-private partnerships aiming at faster innovation processes.

The authors wish to thank the industry representatives from Ruukki Construction Ltd and SSAB Europe Ltd who have been active in planning and supervising the work.

Espoo 3.3.2017

Authors

Contents

1. Introduction

Finite element models (FEM) are becoming common tools for structural steel designers and structural products manufacturers. They are typically used to optimize geometry and material of the product or structure, to predict its loading resistance or to verify the design assumptions. With the introduction of new high-strength steel grades, improved fabrication tolerances and joining methods, it is possible to perform very accurate, highly optimized and complex design assisted by FEM. Such design may, however, require a special attention in particular details where the concentration of stress and strain cannot be avoided. The essential prerequisite to prevent material failure in this situation is sufficient material ductility.

This report presents a FEM approach to perform the following engineering tasks dealing with the structural steel details subjected to tensile stress concentrations:

(a) **To provide safe prediction of tensile resistance of a detail** using simple FEM tools and a basic knowledge of the material behaviour. Several alternative methods are presented in Chapter 4 (Prediction of ductile failure).
Some of the criteria may allow plastic deformation and diffuse necking in the localized areas, and therefore are not suitable for cyclic loading.

(b) **To aid the designers with the selection of steel grade** through the prediction of minimum required material ductility that will ensure safe performance of the studied detail.

The predicted material ductility may be lower than the minimum requirements of the design codes or higher if the exceptional performance is needed (e.g. large elongation of the area with stress concentration without failure). Chapter 5 (Verification of material ductility) presents the description of this method.

Both approaches are based on the same assumptions concerning numerical models described in Chapter 2 (Geometry of the details) and material models described in Chapter 3 (Constitutive models).

The approaches were verified by experiments (including results from previous tests and new experiments) and numerical simulations. However, there are needs to perform broader experimental and numerical studies to extend the applicability area from the restrictions described also in the report.

2. Geometry of the details

The FEM-based methods presented in this report are generally suitable to structural details, where the whole cross-section or its part is subjected to tension that may be equal or higher than the ultimate tensile strength of the material. Such details are prone to diffuse necking in the most loaded areas. The localized stress concentration is typically observed in the cross-sections reduced by holes or notches and welded joints.

The presented approach was verified by a limited number of experiments and numerical simulations. Therefore, it can be used directly only within the given limits as explained in the following sections.

2.1 Geometry prerequisites

The geometry of the structural details suitable for the recommended modelling techniques is now limited to the cross-sections reduced by round holes, corners and/or round notches in the tensile areas. The reason for this restriction is that sharp notches and corners tend to be more sensitive to the mesh density of the numerical models. Although, it is assumed that coarser mesh will lead to outcomes that are more conservative, virtual testing yet did not prove this assumption. One example of welded joint is presented as the case study in this report.

Additionally, the acceptable failure mode of the details in tension should be ductile. The details with very thin plates (high aspect ratio of the individual plates) may also fail in shear or combination of both failure modes. The recommended maximum aspect ratio of the plated elements is therefore 1:8.

2.2 Initial imperfections

The finite element models described in this report are based on the assumption that the diffuse necking is a stability phenomenon caused by non-uniform stress distribution due to imperfect material or imperfect geometry in tension. Since it is convenient to create models with homogenous isotropic material behaviour, it might be necessary to introduce initial geometric imperfections in the critical areas of uniform nominal cross-sections. This will enable the initiation of the numerical instability in the models.

The typical example of the model requiring initial imperfections is a plain coupon subjected to concentric axial tension. The proper magnitude of the initial imperfection is important especially for materials with nearly perfect plastic behaviour.

(a) The imperfection magnitude should be as small as possible, ideally in the order of magnitude of the surface roughness (e.g. 10 to 100 μm). Depending on the FEM solver, necking may not be initiated with very small magnitudes due to round-off error. Therefore, it is recommended to use double precision if possible. On the other hand, in some cases only the mesh irregularities caused by denser meshing in the critical area may be sufficient to trigger the necking.

(b) It is recommended that the critical cross-section has the nominal dimensions, while the rest of the model is slightly larger. This will produce accurate stress and strain distribution in the critical cross-section.

The details with holes and notches, or cross-sections subjected to tension and bending do not require initial imperfections, because their stress distribution is already non-uniform in the early loading phase.

3. Constitutive models

Material parameters for creating a proper stress-strain relationship of structural steel can be obtained from several sources.

(a) **Eurocode 3** (EN 1993-1-1, EN 1993-1-4 and EN 1993-1-12) provide the nominal values of yield strength f_y, ultimate strength f_u and modulus of elasticity E. EN 1993-1-4 recommends also non-linear parameter n for Ramberg-Osgood models of stainless steels, but these models are not discussed in this report. It should be noted that the material parameters obtained from ductility limits for elongations (e.g. uniform strain or elongation at failure) in the Eurocode are too conservative especially for mild steels.

(b) **Declaration of performance** (EN 10025-1) is required in connection to the CE marking of structural steel products, and it contains minimum yield strength, tensile strength ranges and minimum elongations for different thicknesses. This declaration is available on the producer's website, and therefore it is possible to utilize the values in the design phase before the steel is ordered and delivered.

(c) **Material certificate** (EN 10204:2004) contains the test results such as yield strength, ultimate strength R_m and elongation at failure. This certificate is usually delivered with the material.

(d) **Tensile test raw results** (tabular data of measured load and displacement) are the most accurate information for building the constitutive model of the material. They are usually produced directly by the testing machine. They can be transformed into true stress-strain curves and used directly as inputs for the definition of plasticity in the finite element models.

Table 1. Material parameters available from different sources.

Grade	Yield strength	Tensile strength	Uniform elongation	Elongation at failure	Failure load
Eurocode 3	f_y	f_u	$\varepsilon_u \geq 15\ \varepsilon_y$ 1)	$A5 \geq 10$ - 15% 1)	no
Declaration of performance	R_{eH} or $R_{p0,2}$ 2)	R_m 2)	no	$A5$ 3)	no
Inspection certificate	R_{eH} or $R_{p0,2}$	R_m	no	$A5$	no
Tensile test results	yes	yes	yes	yes 4)	yes 4)

1) Minimum values based on ductility limits
2) Ranges of values
3) Minimum values
4) The failure load may not be recorded due to the removal of strain gauges before the failure

3.1 Elastic - ideally plastic model

The basic parameters provided by the Eurocode 3 are not sufficient to utilize strain hardening in the material up to the ultimate strength f_u because the standard does not contain the value of uniform elongation ε_u. Therefore, the most conservative assumption is the ideally plastic material after reaching the yield strength f_y. It should be noted that in tension, the engineering stress (or the tensile resistance) starts rapidly decreasing due to the diffuse necking beyond this point as demonstrated in Figure 1.

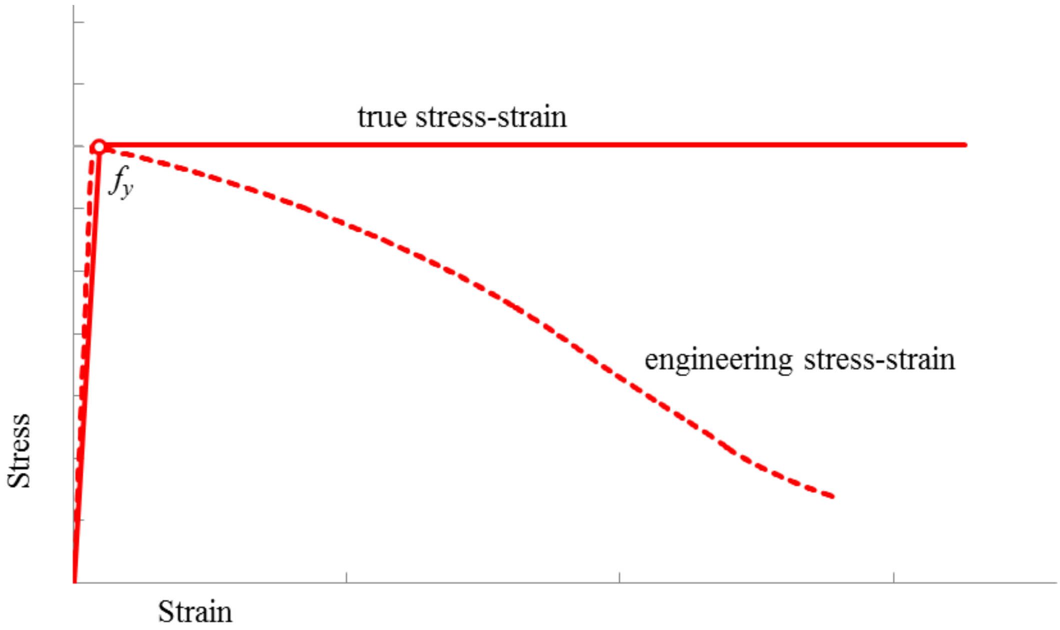


Figure 1. Example of elastic-ideally plastic model behaviour in tension.

The true value of yield stress required by FEM solvers is slightly higher and should be calculated according to the Equation (1).

$$\sigma_t = \sigma(1+\varepsilon) \text{ and } \varepsilon_t = \ln(1+\varepsilon) \quad (1)$$

Where the engineering stress σ is the yield strength f_y in this particular case and engineering strain ε is the yield strain f_y/E.

For instance, for steel S690, the recommended yield strength f_y is 690 MPa and modulus of elasticity E is 210000 MPa. Therefore the true value of yield stress required by the finite element solver is 690·(1+690/210000) = 692 MPa.

3.2 Model with linear strain hardening

Declaration of performance or inspection certificate may provide additional material parameter called the elongation at failure. If the declared elongation is of proportional test specimen (usually for thicknesses 3 mm and higher), it is called $A5$. The knowledge of $A5$ with no information about ε_u enables the utilization of the ultimate strength only partly, but it can still be more valuable than ideally plastic model. The reduced ultimate strength of the model $f_{u,red}$ shall be calculated according to the Equation (2).

$$f_{u,red} = f_y + \frac{f_u - f_y}{A5} \varepsilon_{u,\min} \qquad (2)$$

where $\varepsilon_{u,min} = 15(f_y/E)$ is the minimum uniform elongation required by the Eurocode 3.

It should be noted that such models can achieve slightly higher resistance than the ideally plastic materials, but their ductility in tension is significantly improved because the necking starts after reaching the minimum uniform elongation $\varepsilon_{u,min}$ as demonstrated in Figure 2.

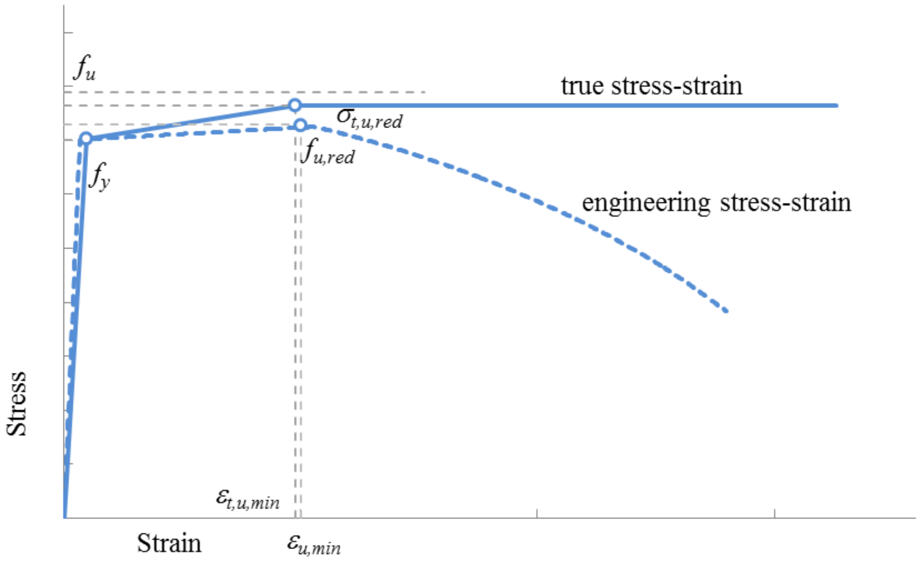


Figure 2. Example of model with linear hardening behaviour in tension.

For instance, for steel S690QL produced by SSAB, the following parameters can be obtained from the declaration of performance according to EN 10025-6:

- Minimum yield strength is 690 MPa;
- Minimum tensile strength is 770 MPa;
- Elongation at failure is 14%.

The minimum uniform elongation $\varepsilon_{u,min}$ required by the Eurocode 3 for this grade is than $15 \cdot (690/210000) = 4.93\%$ (modulus of elasticity E is recommended in EN 1993-1-1). The reduced ultimate strength $f_{u,red}$ can be calculated as $690+(80/0.14) \cdot 0.0493 = 718$ MPa and its true value $\sigma_{t,u,red}$ is $718 \cdot (1+0.0493) = 754$ MPa according to the Equation (1). The true logarithmic strain $\varepsilon_{t,u,min}$ corresponding to this stress is then $\ln(1+0.0493) = 4.81\%$.

Table 2. Examples of SSAB materials and corresponding model parameters.

Grade	*Material parameters provided by the steel producer*				*Engineering model*		*True stress-strain model*	
	$f_{y,min}$ [MPa]	$f_{u,min}$ [MPa]	t [mm]	$A5_{min}$ %	$\varepsilon_{u,min}$ %	$f_{u,red}$ [MPa]	$\varepsilon_{t,u,min}$ %	$\sigma_{t,u,red}$ [MPa]
S235J2 strip	235	360	3 ≤ t ≤ 16	24	1.68	244	1.66	248
S275J2 strip	275	410	3 ≤ t ≤ 16	21	1.96	288	1.95	293
S355J2 strip/plate	355	470	3 ≤ t ≤ 40	20	2.54	370	2.50	379
S690Q/QL plate	690	770	3 ≤ t < 50	14	4.93	718	4.81	754
S890QL plate	890	940	3 ≤ t < 50	11	6.36	919	6.16	977
S960QL plate	960	980	3 ≤ t < 50	10	6.86	974	6.63	1040

3.3 True material model

The tensile test results are usually very accurate based on measurements of load and corresponding elongation of a coupon or bar. They can be easily converted into true stress and true strain curve up until the ultimate load according to Equation (1). However, the behaviour beyond this point depends on the test specimen geometry and the distance between the measuring points, because it involves diffuse necking and the stress and strain is not equally distributed in the cross-section anymore.

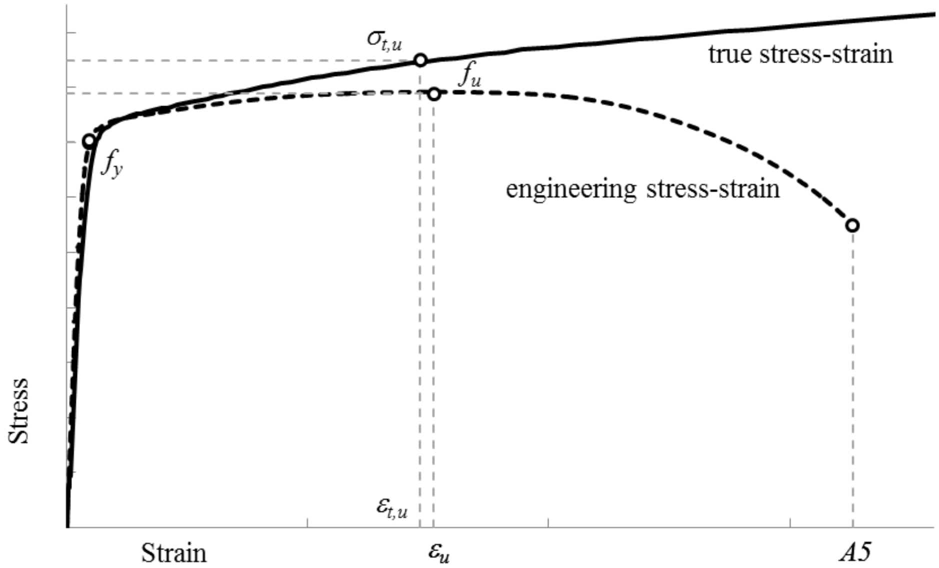


Figure 3. Example of model obtained from tensile test data in tension.

It is possible to obtain true stress-strain relation beyond the onset of necking for instance using different empirical approximation methods or by simple iterative curve-fitting of the FE model of the coupon to match the measured load and elongation. Since the FE model of the coupon is required for the methods presented in the later sections of this report, we recommend this curve-fitting approach.

4. Prediction of ductile failure

In order to provide a safe design or prediction of the loading resistance of the structure, it is essential to ensure that the concentrated internal stress and strain distributions do not cause ductile failure of the material. For that purpose, different design criteria can be used. The most simple failure criteria are given in Eurocodes (Annex C of EN 1993-1-5 and EN1993-1-6). They are based on von Mises equivalent plastic strain or principal plastic strain. The limitations in these standards in their present form are hindering the full utilisation of FEM-based plastic design. In addition, more sophisticated macroscopic damage models with a wide range of experimental parameters are available.

Methods (a) and (b) below can be used for the design according to the design codes. Other methods are suitable for example for optimization of prototype details and shall be verified by testing.

4.1 Simple calculation limits

(a) **5% strain -** The principal membrane strain (at mid-plane of the plate) is limited to 5% in regions subjected to tensile stresses in informative Annex C of EN 1993-1-5 (plated structural elements). The principal structural strain is also limited to 5% in normative Annex B of EN 13445-3 (pressure vessels). EN 1993-1-12 refers to this Annex, and therefore the limit can be used up to S700. It should be noted that it is not practical to use 5% limit in areas with holes, notches or similar stress concentrations, because is leads to overly conservative results.

(b) **50x yield strain -** Plastic limit state (LS1) in EN 1993-1-6 (shell structures) is based on von Mises equivalent plastic strain and the limit is $\varepsilon_{mps} = 50\varepsilon_y = 50f_y/E$. This condition shall be fulfilled at any point not closer to a notch or local discontinuity than the thickest adjacent plate thickness. It should be noted that EN 1993-1-6 is intended for use in conjunction with EN 1993-1-1 (not EN 1993-1-12), and therefore it is applicable only up to S460. Yield strength f_y = 460 MPa results in $50\varepsilon_y$=11%.

Table 3. 50x yield strain limit for common structural steel grades (covered by EN 1993-1-1)

Grade	f_y MPa	ε_y %	$50\varepsilon_y$ %
S235, S235 W	235	0.11	5.6
S275, S275 N/NL/M/ML	275	0.13	6.5
S355, S355 N/NL/M/ML/W	355	0.17	8.5
S420 N/NL/M/ML	420	0.20	10.0
S450	440	0.21	10.5
S460 N/NL/M/ML/Q/QL/QL1	460	0.22	11.0

(c) **Uniform elongation of material -** One conservative approach is to limit plastic strains to the uniform elongation $\varepsilon_{u,}$ of the material. Up to this value, the plastic strain in plain coupons is uniformly distributed in the cross-section, and therefore very similar to the elongation. Minimum value of $\varepsilon_{u,min} = 15\varepsilon_y$ is required in EN 1993-1-1. As an example $15\varepsilon_y$=4.9% for a steel with yield strength of 690 MPa. Similarly to the 5% rule, the practical usability of such criterion is limited to the areas without stress concentration.

Table 4. Minimum uniform elongation $\varepsilon_{u,min} = 15\varepsilon_y$ of material for common structural steel grades (covered by EN 1993-1-1 and EN 1993-1-12).

Grade	*Thickness* *mm*	f_y MPa	ε_y %	$\varepsilon_{u,min}$ %
S235, S235 W	any	235	0.11	1.68
S275, S275 N/NL/M/ML	any	275	0.13	1.96
S355, S355 N/NL/M/ML/W	any	355	0.17	2.54
S420 N/NL/M/ML	any	420	0.20	3.00
S450	any	440	0.21	3.14
S460 N/NL/M/ML/Q/QL/QL1	any	460	0.22	3.29
S500 Q/QL/QL1/MC	≤ 50	500	0.24	3.57
	≤ 100	480	0.23	3.43
	≤ 150	440	0.21	3.14
S550 Q/QL/QL1/MC	≤ 50	550	0.26	3.93
	≤ 100	530	0.25	3.79
	≤ 150	490	0.23	3.50
S600 MC	≤ 16	600	0.29	4.29
S620 Q/QL/QL1	≤ 50	620	0.30	4.43
	≤ 100	580	0.28	4.14
	≤ 150	560	0.27	4.00
S650 MC	≤ 8	650	0.31	4.64
	≤ 16	630	0.30	4.50
S690 Q/QL/QL1	≤ 50	620	0.30	4.43
	≤ 100	580	0.28	4.14
	≤ 150	560	0.27	4.00
S700 MC	≤ 8	700	0.33	5.00
	≤ 16	680	0.32	4.86

(d) **Elongation at failure -** A less conservative approach is to limit true plastic strains to the value of elongation at failure $A5$**.** Since the real plastic strain becomes much higher than the elongation after the onset of necking, this criterion appears to be a safe approximation applicable also in the areas around the holes and notches. This limit was verified on the literature study made for test data of notched test specimens.

4.2 Damage prediction models

(e) **SMCS model (Stress Modified Critical Strain) –** SMCS is the simplest of the macroscopic damage models. The damage curve, described by the critical equivalent plastic strain, is a monotonic decreasing function

$$\varepsilon_{eq} > \varepsilon_{cr} = \alpha \cdot e^{-1.5T} \quad (3)$$

where stress triaxiality T is the ratio of the hydrostatic stress and von Mises stress. Exponent 1.5 is commonly used in the expression for steels. However, without experimental verification it can provide unconservative results. The toughness parameter α for steels is in range of 1 to 5. Then, depending on the material grade, the critical plastic strain results in 22%–112%, if $T = 1$. The parameter α is usually calibrated on tests and FEM analyses for smooth-notched CNT (circumferential notch tensile) specimens.

SMCS model can provide accurate fracture predictions for many practical conditions, such as the necked ligament between bolt holes, the necked cross section of an un-notched cylindrical bars, structural moment connections, or circumferential notch tensile (CNT) specimens. In these cases, fracture typically initiates internally, where the stress triaxiality is relatively high (T>0.75) and then propagates outwards towards the surface of the material. However, there are other situations where fracture may initiate on the surface of the material, where triaxiality is typically lower (T=0.33–0.75), and then propagates inward. Fracture initiation on the surface has been observed for example in large scale tests on structural braces and column base plate tests.

(f) **Complex models -** More sophisticated macroscopic damage models are usually used for simulations of sheet forming or car deformations in accidents. A large number of tests for different specimens and complementary FEM analyses need to be done. These damage models are often laborious and difficult to interpret in practical design.

(g) **Plastic strain at coupon failure elongation -** Force-displacement curve until $A5$ elongation is always necessary in material testing, if real material properties are used in analysis of the critical detail. Then the critical plastic strain corresponding $A5$ can determined by modelling the tensile test specimen as well as the detail. This critical strain can be used as a damage criterion for details with holes and notches where the failure is initiated on the surface. The safe application area of this method can be extended to the other details by experiments and supplementary FEM calculations.

The last method (g) from this overview proved to be the best combination of accuracy and simplicity because it requires only standard experimental tests results (plain coupons of uniform cross-section), but still provides reasonable results in FEM simulations of structural details with stress concentration. This prediction is further explained in the following sections.

5. Verification of material ductility

The calculation method presented in this section predicts the minimum required elongation at failure of tensile coupon $A5$ to fulfil the selected design resistance of a structural detail. Materials that do not satisfy one or more of the ductility requirements of the Eurocode [1][2] can be utilized here to check how large elongation at failure $A5$ is needed for the combination of their material parameters f_u/f_y and ε_u in a particular design situation.

The method relies on a standard material coupon test and two numerical models, one model of the detail and one model of the coupon itself. The knowledge of exact stress-strain curve of the material and the geometry of tested coupon is not essential, but is recommended because it brings less conservative results. The method can be, however, used only with the basic material knowledge of f_y, f_u, E and elongation $A5$ (if it is known from the mill certificate or declaration of performance according to the harmonized technical specification EN 10025-1).

5.1 Numerical models

Two 3D finite element models have to be created in order to use the verification described in this section. Both models shall be composed of 3D deformable elements (either bricks or tetrahedrons).

(a) **<u>The model of the tensile coupon</u> -** The model should represent the real geometry of the test. The surface imperfections of the real coupon should be smaller than the imperfections implemented in FEM model of the coupon. This ensures higher stress concentration in the model, and therefore more conservative results.

(b) **<u>The model of structural detail</u> -** The finite element mesh of the structural detail should have at least the same density as the mesh of the simulated coupon test. It is recommended that the same software be used for both numerical models with the same element types and calculation settings.

5.2 Required elongation at failure

The prediction of required ductility is based on the knowledge of a single parameter, the maximum equivalent plastic strain ε_{eq} in the critical cross-section of the tensile test. As long as the following conditions are satisfied, it is safe to assume that the minimum required elongation at fracture of coupon $A5$ is the elongation when equivalent plastic strains ε_{eq} reach the same level in structural detail in a particular design situation:

(a) **<u>Stress triaxiality at the failure of the structural detail is smaller than the stress triaxiality of tested coupon.</u>** This condition is true for structural details with notches and holes in tension and plain rectangular coupons except for the small plastic deformations.

(b) **<u>The relation between failure strain and stress triaxiality (the damage curve) of a material is monotonic decreasing function.</u>** This is also true for the common structural steels and the details prone to ductile failure in tension.

The calculation is based on the assumption that strains larger than the uniform elongation of tensile coupon ε_u can be accepted in localized areas of statically loaded structures. In such cases, diffuse necking may develop in localized areas. However, the load should not cause ductile failure in materials, and therefore the plastic strains and hydrostatic stress (represented by stress triaxiality) should remain within a given range. Unfortunately, it is impossible to describe exactly the relation of stress triaxiality and plastic strain at material failure with data from commonly used coupon test. Therefore, the present method involves certain conservativeness in the generated results. If a more accurate solution is needed, it has to be based on rather complicated testing programme in combination with more complex damage prediction models.

The entire process is described in Figure 4, Figure 5 and the following paragraphs.

(a) **Selection of the design limit** - the design limit can be expressed as load (e.g. ultimate load), deformation (e.g. 3 mm elongation) or any other measurable property of the detail. Many existing limits in the Eurocodes can be utilized here. Depending on the limit, several suitable material models can be selected.

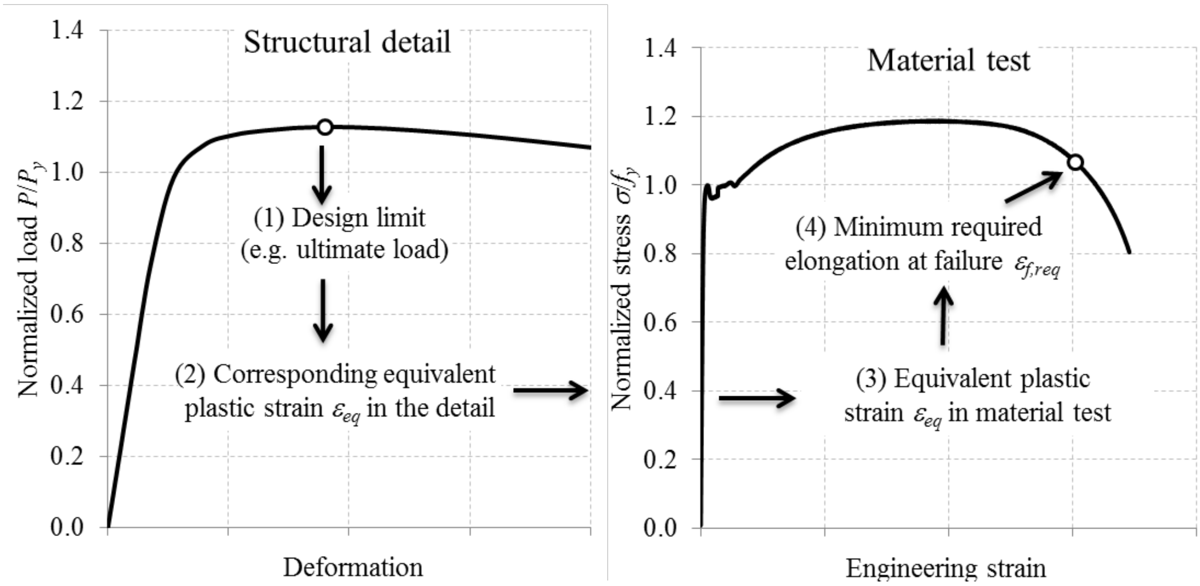


Figure 4. Prediction of minimum required elongation at failure based on the FEM model of the structural detail (left) and FEM model of the coupon test (right).

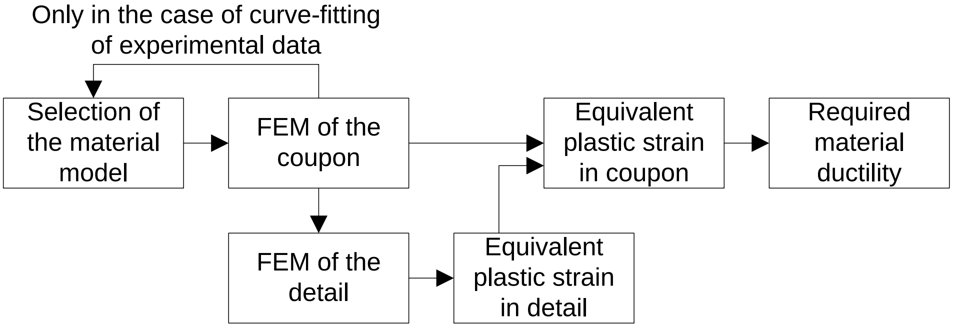


Figure 5. Verification of material ductility.

(b) **Selection of material model** - Based on the available knowledge of the material's parameters, one of the possible constitutive models can be selected from Chapter 3. The yield strength of the material is required as a minimum input, but for some design limits (e.g. the ultimate load), the knowledge of more material parameters might be needed.

More complex models that are based on the Declaration of Performance, Inspection Certificate or real tensile testing provide usually the knowledge of material's elongation at failure $A5$, and therefore the purpose of the calculation would be to verify if it is sufficient in the current design situation.

Generally, the safe assumption beyond the ultimate load is ideally plastic behaviour of true stress-strain curve. However, better performance can be achieved with the knowledge of complete coupon testing data (or at least the load and elongation at failure), because the true stress-strain behaviour can be curve-fitted to match this data.

(c) **Finite element model of the standard coupon -** If the material model is based on the particular coupon test or is associated with the particular sheet/plate thickness, the dimensions of the coupon should be selected accordingly. Otherwise, the rectangular coupon with the material thickness corresponding to the studied detail and aspect ratio at least 1:4 (but not too high to cause shear failure) is recommended.

The coupon mesh should be denser in the area of diffuse necking, but its density can be based on the computational capacity of the FEM solver. Generally, denser mesh results in less conservative prediction of minimum required elongation at failure. Mostly it is sufficient to simulate only 1/8 of the coupon due to its symmetric behaviour (see Figure 6).

Plain coupons might require initial imperfections in order to initiate diffuse necking in the middle of the coupon (see Chapter 2.2). The cross-sectional area can be then reduced locally at least by the value of the surface roughness. Higher reduction will result in less conservative prediction.

FE model should not contain any provision for the failure simulation and should be able to provide results also beyond the ultimate load. Therefore, it is recommended to load the model with deformation that should be larger than the expected failure elongation.

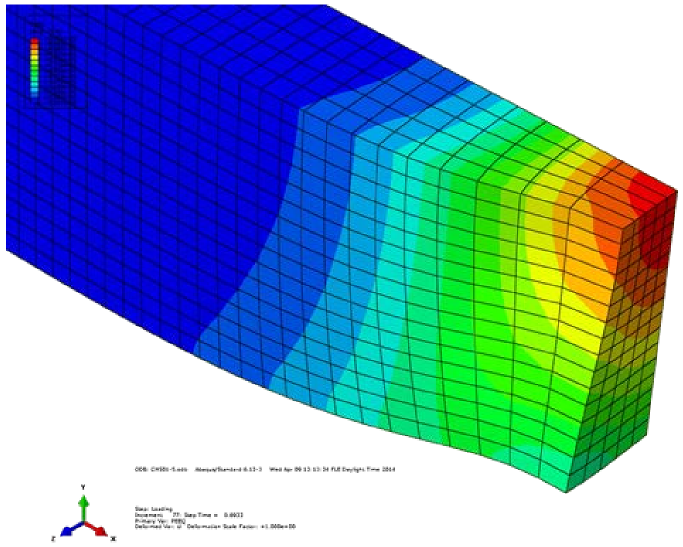


Figure 6. Simple FE model (3 symmetry planes) of the coupon with the largest strain in the middle of the crosss-section.

(d) **Finite element model of the studied detail -** The same material model and element type should be used in the FEM model of the detail. It is also recommended to use the same FE solver with the same basic settings.

The meshing recommendations are based on the FEM model of the coupon, but the initial imperfection might not be necessary because of the presence of holes or notches (see Figure 7).

Similarly as the coupon, this FE model should not contain any provision for the failure simulation and should be able to provide results also beyond the ultimate load. Therefore, it is recommended to load the model with deformation that is large enough to cover the required design limit(s).

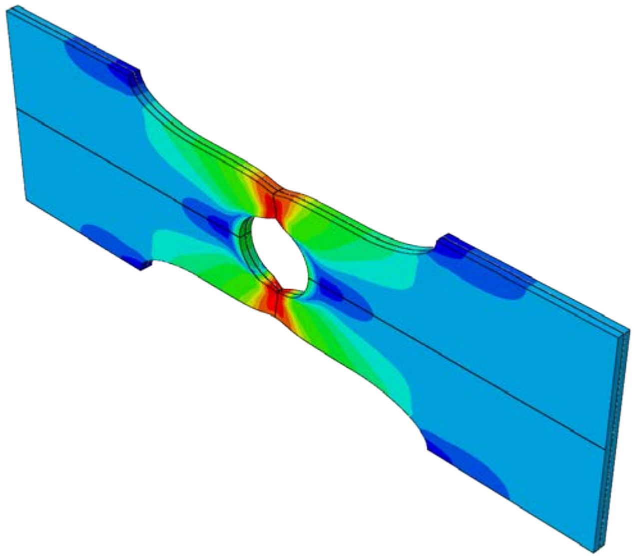


Figure 7. FE model of the detail with the largest strain at the edge of the hole.

(e) **Determination of equivalent plastic strain at the design limit in detail FE model** - The maximum equivalent plastic strain of the model should be obtained at the load level corresponding to the selected design limit. The largest strains are usually at the edge of holes and notches.

(f) **Determination of the corresponding elongation of coupon FE model -** After the same equivalent plastic strain is reached in the coupon model, it is assumed that ductile failure can be allowed in the material. The elongation at this point is considered to be the required minimum elongation at failure $A5$.

This prediction can be too conservative in some cases, because in reality also coupons failing at lower strain levels might be capable of reaching the desired strain in the detail.

The location of maximum equivalent plastic strain in the coupon model is in the middle of the cross-section.

If the FE model of the coupon or the detail starts necking along any longer path than the smallest net-section of the model, it might be indication of the shear failure that is not yet applicable for this method.

6. Solved numerical examples

6.1 Centre hole in tension

As an example, simple numerical model of plate 8x80 mm with 8 mm centre hole in tension was selected. The length of the FE model of the plate is 100 mm and it is loaded with deformation. In reality, only ¼ of the model was created with the appropriate symmetry boundary conditions.

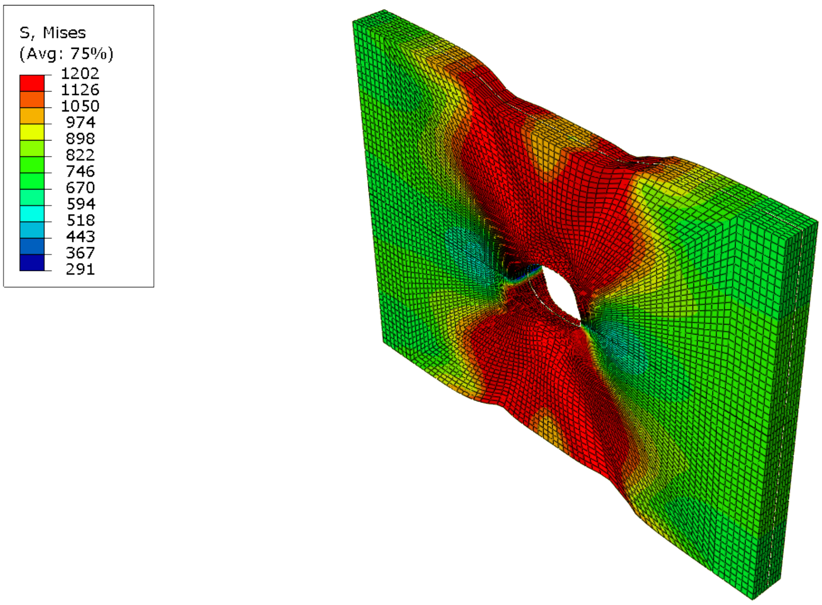


Figure 8. FE model of the central hole in tension (CHT) plate.

The calculation results are verified by the tensile test of the dogbone specimen, in which the central part is identical to the FE model.

6.1.1 Material models

Four constitutive models of high-strength steel S960 were selected to study the recommended calculation limits:

- **Elastic-ideally plastic model** with $f_y = 960$ MPa.
- **Model with linear strain hardening** based on the Declaration of Performance by SSAB (see Table 2), where $f_y = 960$ MPa, $\sigma_{t,u,red} = 1040$ MPa, and $\varepsilon_{t,u,min} = 6.63\%$.
- **Model with linear strain hardening** based on the stress-strain curve fitted to the real coupon test, where $f_y = 1064$ MPa, $\sigma_{t,u} = 1202$ MPa, and $\varepsilon_{t,u} = 2.82\%$.
- **Model with nonlinear strain hardening** fitted to the real coupon test

Their modulus of elasticity is 210 GPa, Poisson’s ratio is 0.3, and the full true stress-true logarithmic strain relationship is in Figure 9.

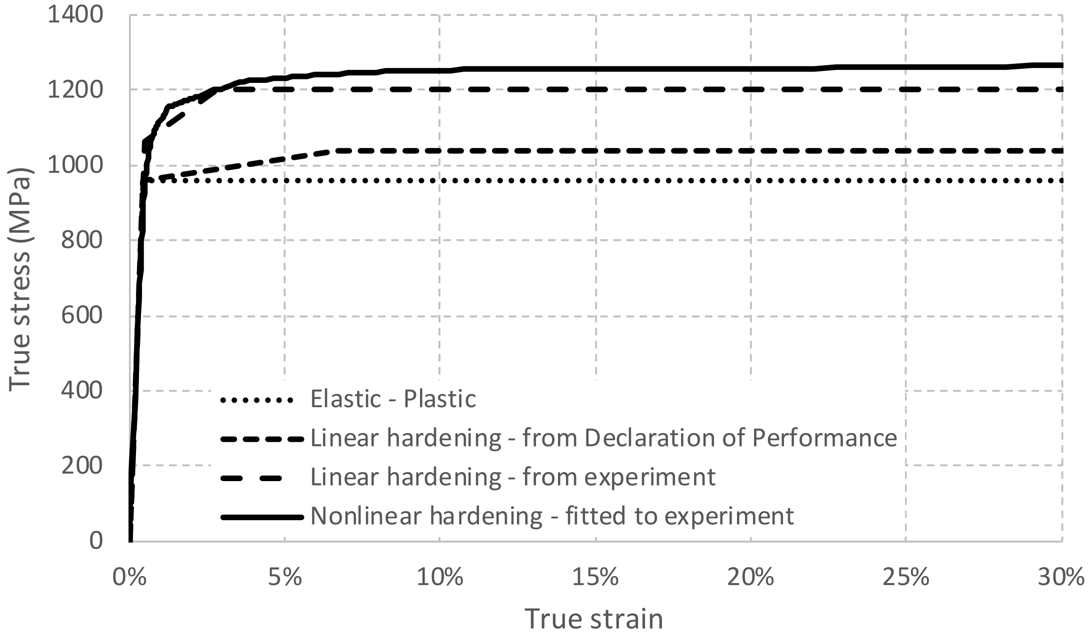


Figure 9. Selected constitutive models.

6.1.2 Estimation of ductile failure

In order to predict safe calculation limit before the real ductile failure of the material, several criteria were selected for the maximum allowable equivalent plastic strain

- **5% strain**
- **50x yield strain** (22.9% equivalent plastic strain)
- **Uniform elongation** 15x yield strain (6.9% equivalent plastic strain) for the first two material models, where the real uniform elongation is not known or the measured uniform elongation 3% for the models based on coupon test.
- **Elongation at failure** 10% for the first material model, where the A5 value is not known or is given by the Declaration of Performance by SSAB, or 9.8% for the models based on coupon test.
- **Plastic strain at coupon failure elongation** 98.2% was obtained from the simulation of the coupon test, and therefore this limit is used only in combination with nonlinear strain hardening material model.

The results of combination of four material models and four strain limits are presented in Figure 10 and Table 5. The failure force F_f and elongation ΔL_f at the failure of the detail (and its % of initial length L_0) are presented in the table as well as the maximum load reached before failure (as % of F_y).

Table 5. Limiting load and deformation in the studied cases.

Material model	Strain limit	Max. load F_{max} [kN]	Failure load F_f [kN]	Elongation ΔL_f [mm]	% of L_0
Experiment		672	596.1	4.31	4.31%
Elastic - Plastic	5% strain	553	549.3	0.52	0.52%
	50x yield strain	553	542.2	1.07	1.07%
	Uniform elongation	553	549.1	0.58	0.58%
	Elongation at failure	553	548.3	0.68	0.68%
Linear hardening (based on DoP)	5% strain	561	554.3	0.53	0.53%
	50x yield strain	561	550.1	1.10	1.10%
	Uniform elongation	561	554.5	0.59	0.59%
	Elongation at failure	561	554.0	0.72	0.72%
Linear hardening (based on coupon test)	5% strain	645	644.8	0.62	0.62%
	50x yield strain	687	687.3	1.37	1.37%
	Uniform elongation	626	626.0	0.53	0.53%
	Elongation at failure	665	664.6	0.84	0.84%
Nonlinear hardening (based on coupon test)	5% strain	64	640.8	0.70	0.70%
	50x yield strain	690	690.4	1.48	1.48%
	Uniform elongation	614	613.8	0.58	0.58%
	Elongation at failure	673	672.9	0.94	0.94%
	Plastic strain at failure elongation	693	645.6	4.46	4.46%

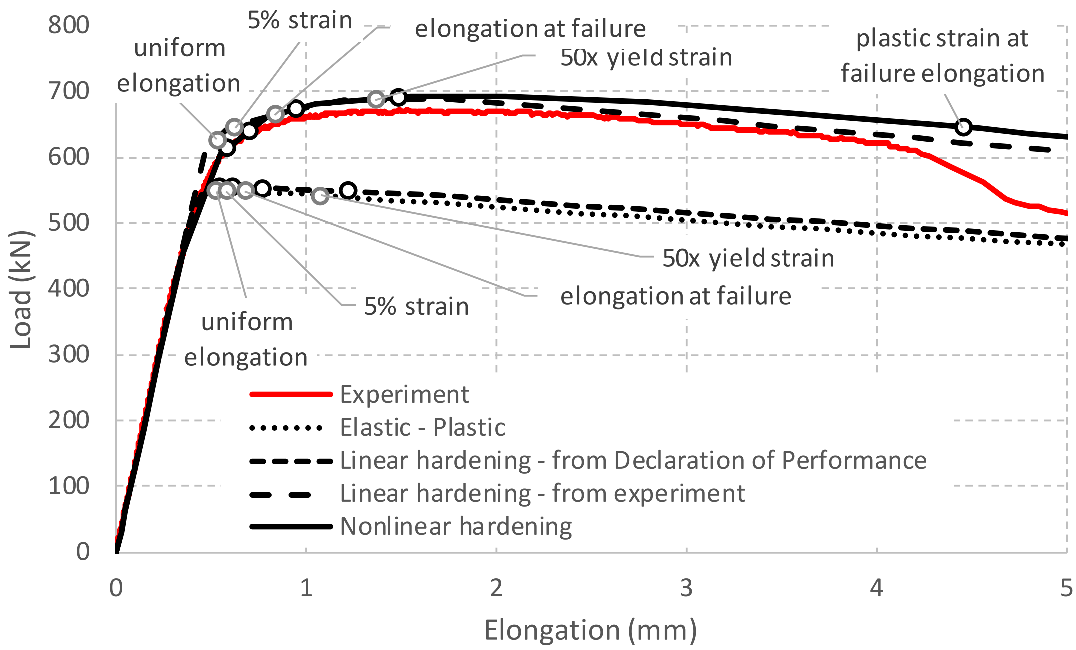


Figure 10. Load-displacement relationship using different material models and calculation limits.

The following results can be observed

- The material models based on the real testing data can offer higher loading capacity (up to 124% of F_y in our case) and deformation capacity (up to 1.48% in our case) within the given limits.
- All of the damage prediction limits had a certain level of conservativeness in this case.
- The limit based on uniform elongation guarantees that the necking is not developing in the detail at all as well as the 5% limit for steels with $f_y \geq 700$ MPa[1]. Such condition might be suitable for cyclic loading but in the statically loaded cases it can be, however, too limiting. The elastic-ideally plastic model is not suitable for the simulation of the details, where the necking should be restricted, because the diffuse necking develops at much lower strains than in reality.
- The limits based on 50x yield strain and elongation at failure allow the localized necking as well as the 5% limit for steels with $f_y < 700$ MPa. This condition does not mean automatically that the ultimate capacity of the detail is reached.

6.1.3 Verification of the material ductility

The second task that can be demonstrated on the same model is the calculation of required elongation at failure of the coupon in the standard testing. For this reason, another FE model has to be produced; the model of the coupon. The coupon model was in this example already created to curve-fit true stress-strain relationship to the real experiment (see Figure 11)

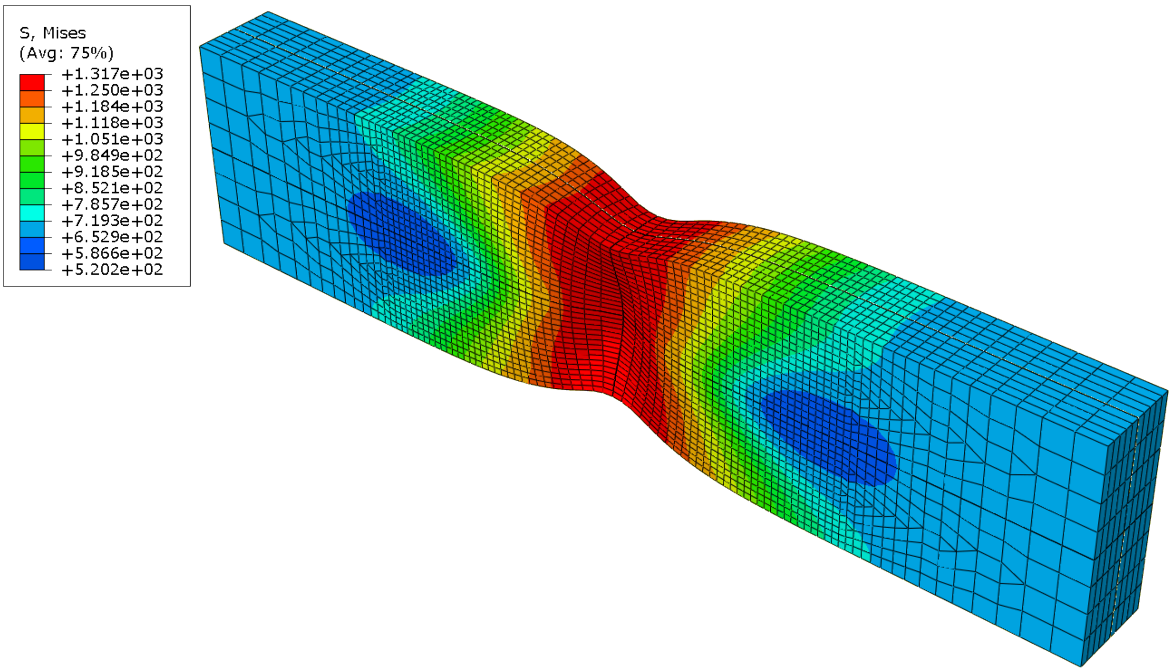


Figure 11 Finite element model of the tensile coupon

The goal of this example is to predict the minimum required elongation at failure $A5$ of the steel S960 to reach the ultimate tensile load of the detail in the net-section. Therefore, the only material models that are able to simulate the ultimate strength of the material will be used.

Table 6. Calculation of the required elongation at failure of the coupon $A5_{req}$

[1] The strain at onset of necking (uniform elongation) should be larger than $15f_y/E$ which is higher than 5%, when $f_y \geq 700$ MPa.

Material model	Ultimate load F_u [kN]	Plastic strain at the ultimate load $\varepsilon_{eq,u}$	Deformation of the coupon at the same strain level ΔL [mm]	Required elongation at failure $A5_{req}$
Linear hardening (based on coupon test)	593	36.5%	1.69	5.6%
Nonlinear hardening (based on coupon test)	589	34.9%	1.79	6.1%

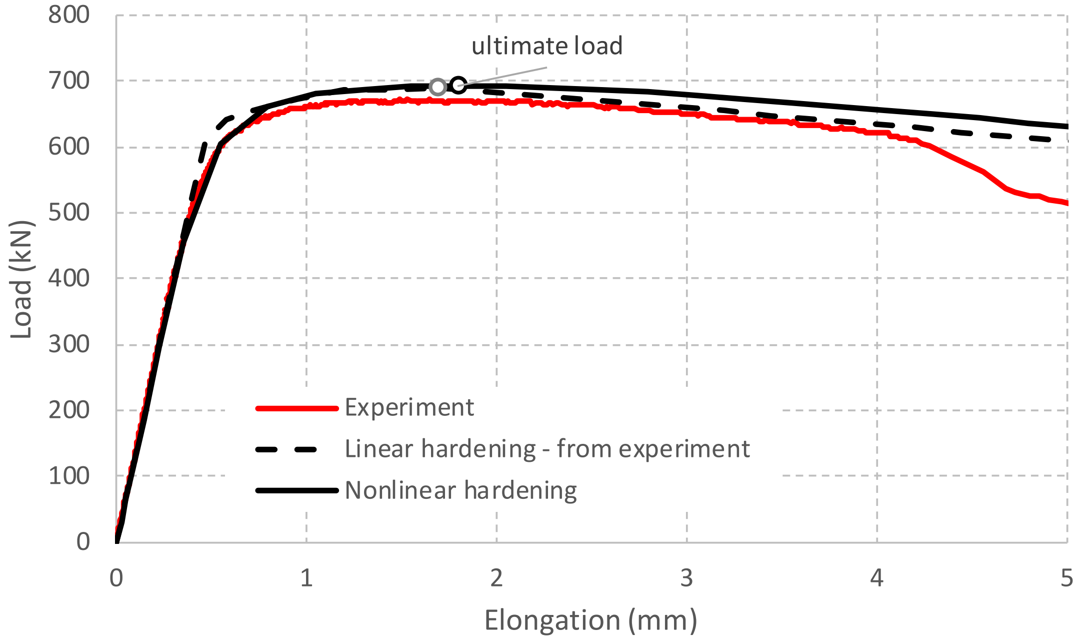
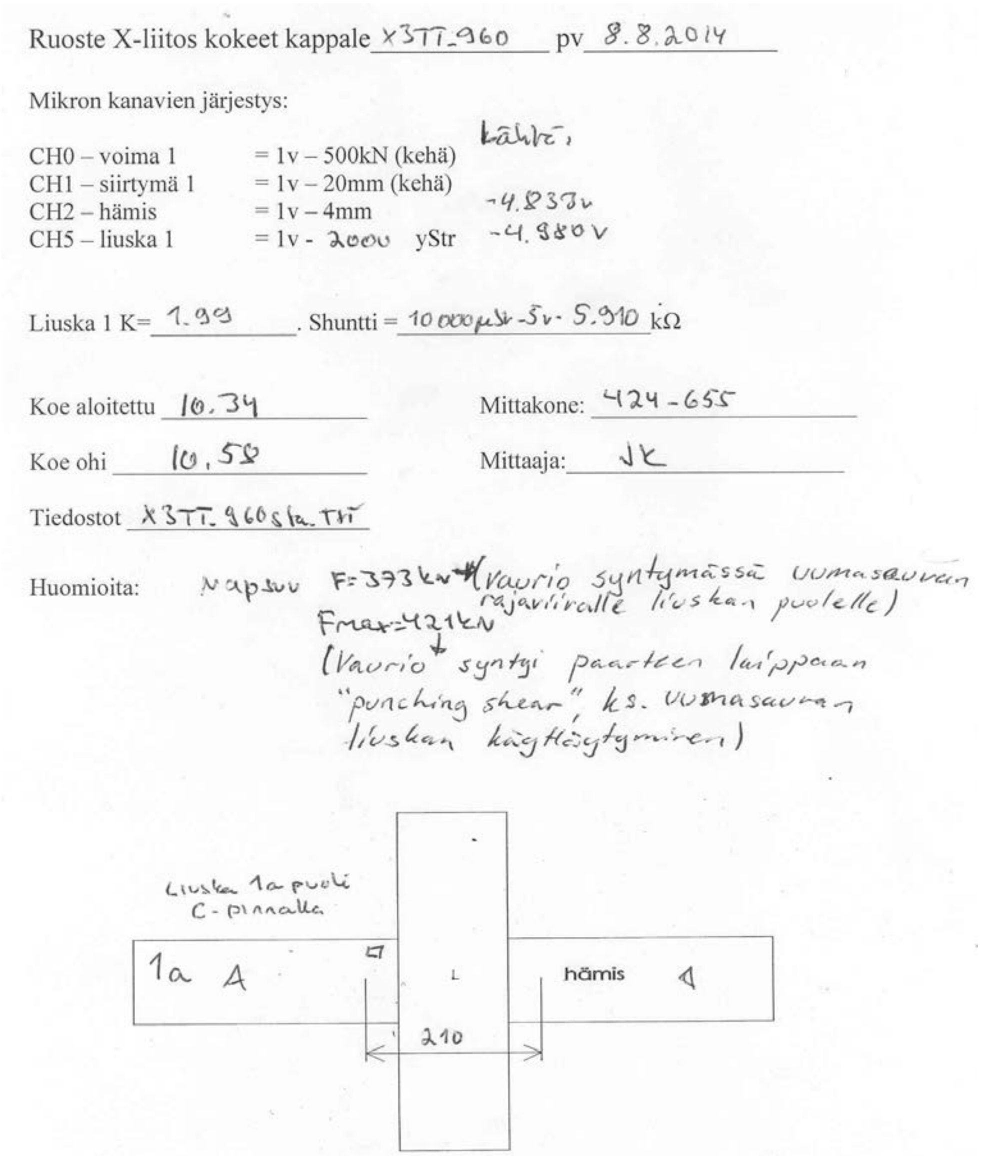


Figure 12 Design limit (ultimate load) in the selected detail

This example shows that the elongation at failure $A5$ has to be higher than 6.1% in order to reach the ultimate load F_u of the selected detail. In addition, the nonlinear hardening model is needed for such prediction, because the simplified linear hardening model provided less conservative results.

6.2 Tubular connection

An attempt was made to use the methodology developed here for predicting the behaviour and failure load for the welded tubular joint X3TT_960. We had access to limited information on the test, and especially the properties of the materials. The lab-test report for the specimen is presented in Figure 13.


Ruoste X-liitos kokeet kappale X3TT_960 pv 8.8.2014

Mikron kanavien järjestys:

CH0 – voima 1	= 1v – 500kN (kehä)	lähtö,
CH1 – siirtymä 1	= 1v – 20mm (kehä)	
CH2 – hämis	= 1v – 4mm	-4.833v
CH5 – liuska 1	= 1v - 2000 yStr	-4.880 V

Liuska 1 K= 1.99 . Shuntti = 10 000µS -5v- 5.910 kΩ

Koe aloitettu 10.34 Mittakone: 424-655

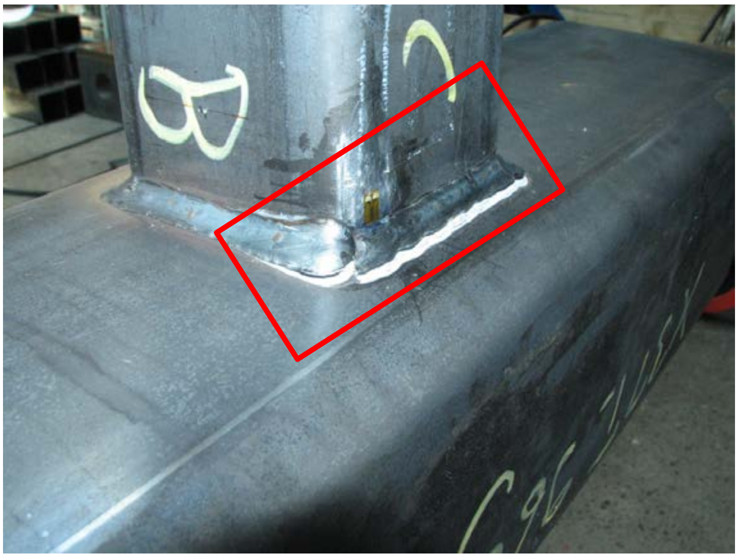
Koe ohi 10.58 Mittaaja: JK

Tiedostot X3TT_960sla.TRT

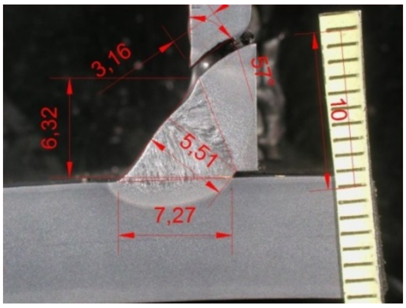
Huomioita: Napsuu F=373kN (Vaurio syntymässä uumasauvan rajaviivalle liuskan puolelle)
Fmax=421kN
(Vaurio syntyi paarteen laippaan "punching shear", ks. uumasauvan liuskan käyttäytyminen)

Figure 13 Lab-test report for X3TT_960, a typical X joint test used in tubular structures

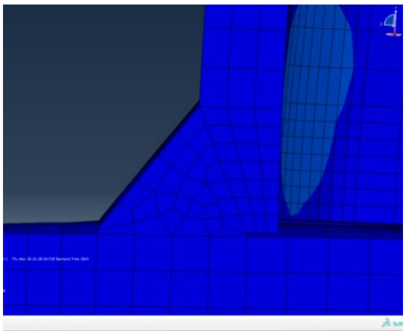
As shown in Figure 14.a, the failure of the joint occurred in the weld, possibly initiating from the heat-affected zone. The relatively prominent weld termination/start in the forefront of the picture may also have played a role in the failure initiation. The weld dimensions of an identically welded specimen were also available from measurement (Figure 14.b). It has to be noted that the welds in Figure 14.a and Figure 14.b are not the same, so there is a degree of uncertainty concerning the dimensions of the weld in specimen X3TT_960. The Finite element rendering of the weld is presented in Figure 14.c.



a)



b)



c)

Figure 14 Failure mode of X3TT_960 initiated in the weld or heat affected zone of the weld. The weld start/termination introduced a prominent discontinuity (and weakness) in the weld.

6.2.1 Material models

Since material tests for the specimen were not available, we used material data from the same producer. The Finite element model was using homogeneous material properties, disregarding the variation of properties in the weld and HAZ. Four material options were used for the models were used in the models (Figure 15).

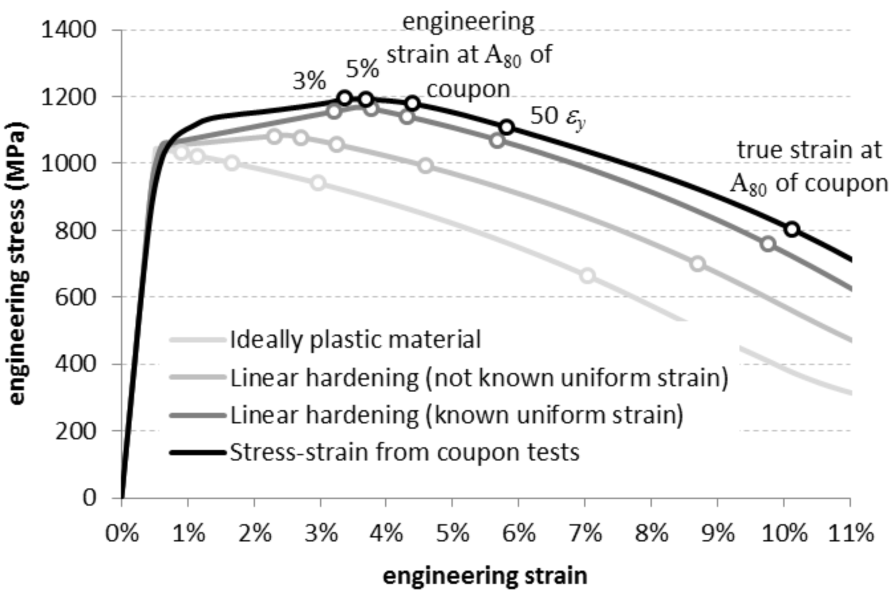


Figure 15. Material models used in the FE models

6.2.2 Estimation of ductile failure and verification of the material ductility

As shown in Figure 16, the FE model is able to pinpoint the stress concentrator responsible for the failure of specimen X3TT_960. The consecutive concentration of plastic strains is also replicated.

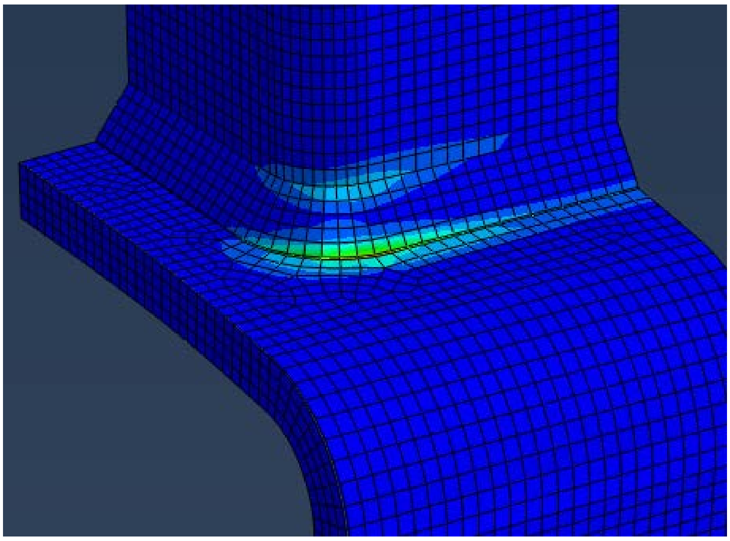


Figure 16. Plastic strains PEEQ in the weld corner region.

When it comes to the behaviour of the joint, the initial stiffness is predicted with good accuracy (Figure 17). The stiffness obtained by FEM starts to depart from the experimental stiffness at higher loads. This can be attributed to ignoring in the models of the weld and HAZ. Due to the weaker material in these regions, the stress and strain concentration in the weld corner will be even stronger this FE model will suggest. Hence, it is expectable that the model will show more stiffness than the experiment. The largest plastic strains (PEEQ) in the FE model are also presented in the Figure 17 on the performance curve. It can be noticed that, plastic strain in the FE model stand at between 9.8-25%, when failure occurred in the experiment. Hence, basing failure prediction on 5% plastic strain will severely underestimate the performance of the specimen to about 220-250kN.

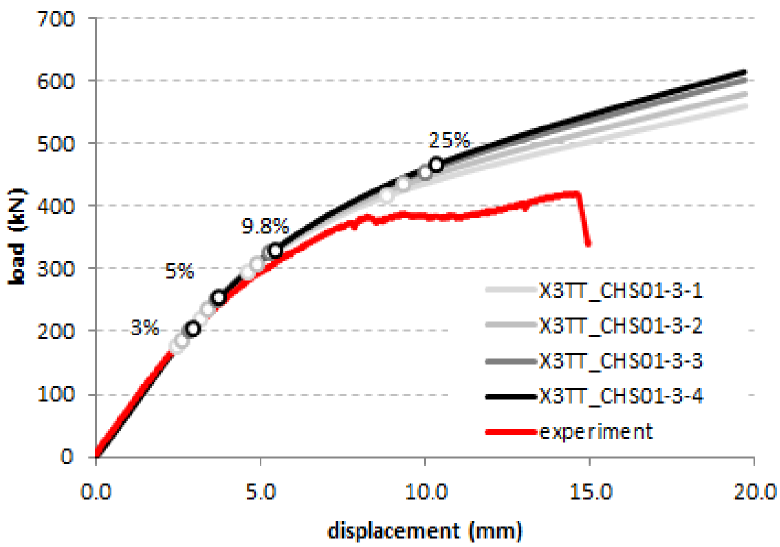
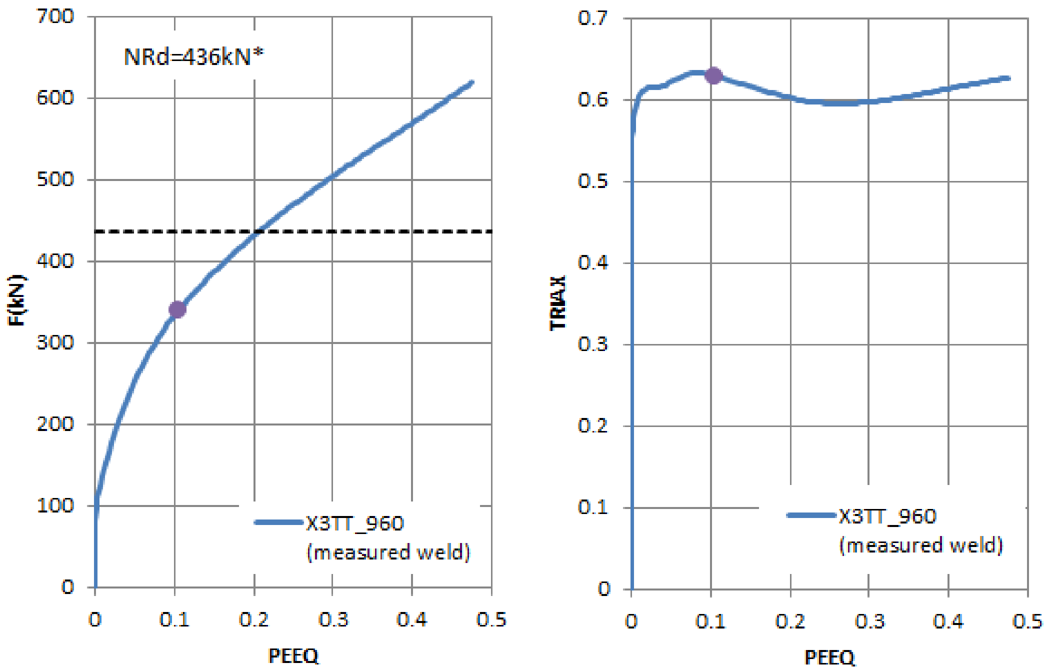


Figure 17. Performance curve of the specimen from experiment (red) and FEM with different material model options (grey/black).

Figure 18 presents the performance curve obtained by FEM, together with the failure load predicted by the EN 1993-1-8 equations (N_{Rd}=436kN). It can be noted that this force level is somewhat larger than the 420kN received in the test (Figure 17). In terms of plastic strain, with a choice of limiting PEEQ at 10% the capacity of the joint would be (under)predicted to 338kN, with PEEQ at 20%, the force estimate would result realistic at 434kN and with PEEQ 25% the capacity would be (over)predicted as 469kN.



(a) (b)

Figure 18. (a) Performance curve of X3TT_960 obtained by FE model and (b) development of the plastic strain and triaxiliaty in the critical element

6.2.3 Conclusions

The welded joint application example of the proposed methodology was unsuccessful. This can be attributed to the model simplifications, especially what concerns the weld and HAZ properties. If these locations are weakened, when compared to the rest of the model, it is probable that stresses and plastic strains concentrate in these regions, resulting in larger localised demands. Unfortunately, for this application not even the base material properties were available, let alone weld and HAZ properties.

For further calibration of the methodology, the use of well-documented tests with known properties of the different components should be used. The modelling approach also has to be a degree more sophisticated, including for instance residual stresses that were ignored here. However, in this case modelling would require more resources than available in this project.

7. Summary

The two methods demonstrated in this report are simple approaches to the problems with the details subjected to the concentrated tensile stresses in some areas or the whole cross-section. They provide insight into the basic questions whether the Eurocode's material requirements are correct or not, could they be eased or not, and how much they could be eased. The methods are validated against real experiments.

This report can serve as a guidance for the finite element modelling of structural details that are subjected to tension, with the risk of initiation and development of diffuse necking.

References

[1] EN 1993-1-1 Eurocode 3: Design of steel structures - Part 1-1: General rules and rules for buildings, European Committee for Standardisation, 2005.

[2] EN 1993-1-12 Eurocode 3: Design of steel structures - Part 1-12: Supplementary rules for high-strength steels, European Committee for Standardisation, 2007.

[3] Dunaud M. Ductile fracture at intermediate stress triaxialities: Experimental investigations and micro-mechanical modelling, Massachusetts Institute of Technology, 2013.