Software for Architecture,
Engineering and Construction

CYPE Connect

Calculation Report

Modelling and analysis of connections for steel structures. This application is integrated into the Open BIM workflow via the BIMserver.center platform.

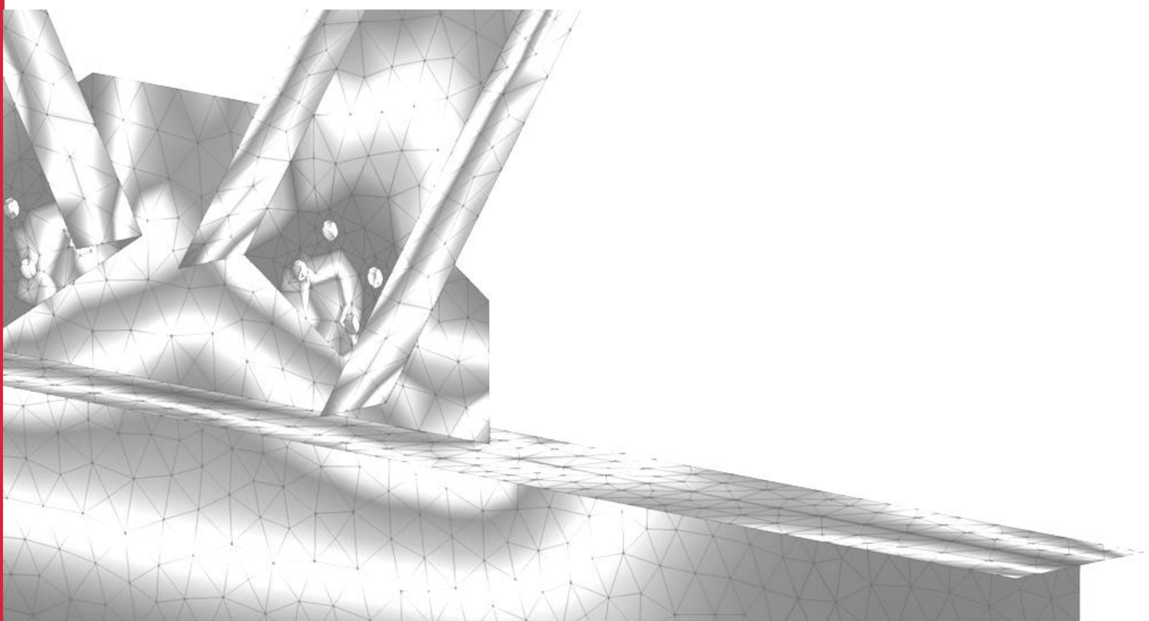


Table of Contents

1 Analysis

1.1 Introduction

Designing connections is one of the most complex issues in structural engineering. Over the years, the usual way of dealing with this problem has been through the use of simplified loadcases and experience, through hand calculation or spreadsheets. A short time ago, there was no legal need to validate dozens or even hundreds of load combinations, and safety factors were higher than they are today.

Currently, there are more design restrictions, and safety factors have been reduced, making design accuracy a crucial aspect.

The traditional way of designing connections can be useful in certain situations, but nowadays we face more complex designs.

When the structure is measured using a bar model, a detailed model of the connections within the structure is not created. To design the connection correctly, users must generate a connection model that respects the actual layout of the connection and also considers the forces applied on the ends of the elements within the connection which are obtained from the resolution of the structure design.

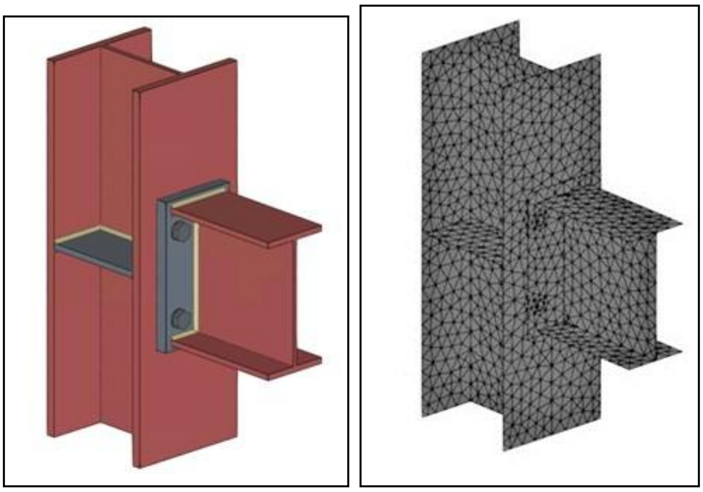
Through the use of the Finite Element analysis, it is possible to address this issue with greater efficiency and accuracy.

CYPE Connect is a tool that allows users to generate models of steel structure connections based on Finite Elements including their analysis and verification according to standard requirements, with minimal or no user intervention, using the world-renowned Finite Element software, OpenSees [1], as its calculation engine.

In the finite element models generated by CYPE Connect, three main elements are recognised: plates, welds, and bolts.

Plates are all the flat elements (the thickness dimension is much smaller than other dimensions) that are involved in the structural model of a connection, such as, the flat elements making up the steel sections, the plates that are used for bolted connections, stiffeners, etc. In figure 1.1, a comparison between the real geometric model of the connection and the discretised model of the connection can be observed. This plate model will be described in more detail in section 1.2.

The bolts and welds make up the model's connection elements and their characteristics will be described in more detail in sections 1.3 and 1.4.



(a) Real geometrical model of the connection. (b) Discrete model of the connection.

Figure 1.1. Discretisation of plates using Finite Elements.

1.2 Plate model

In structures that have a relatively smaller thickness than the rest of the dimensions, as is the case of the plates forming steel structures, *Shell* elements are a good solution (section 29.6.1 in [2]). The used element will be a three-node triangular *Shell* element NLDKGT [3].

These elements consider membrane behaviour (plane stress, compression, shear, and torsional moment) and plate behaviour (out of plane moment). In a non-linear range, plate behaviour is modelled using layered sections. Plate thickness is divided into a number of layers (5 in this case) where the problem to be solved is plane stress. Bending moment analysis is carried out by adding the effects of each layer and it is no longer possible to consider that the stresses in plate thickness are obtained by adding the effects of the membrane and the bending behaviour as it would occur in a linear analysis (section 9.2.4 in [4]).

For finite elements, in order to analyse the necessary integrals, if material nonlinearity is activated, numerical integration techniques are used. If material nonlinearity is activated, the numerical integration necessary to calculate the stiffness matrix is not performed exclusively on the surface (where the Gauss points are used) but is also performed on the thickness.

This plate model will be used for discretising flat elements making up the steel sections and for discretising other plates intervening in the structural model, such as the plates used to make bolted connections or stiffeners. In short, this plate model will be used for discretising any flat element intervening in the structural model of the connection.

The most common material diagrams used in modelling structural steel finite elements are the ideal plastic models or the ideal elastic models with hardening due to deformation. The constitutive law selected for plates and sections will be a bilinear constitutive law with a slope in the plastic section tan−1 (E/1000).

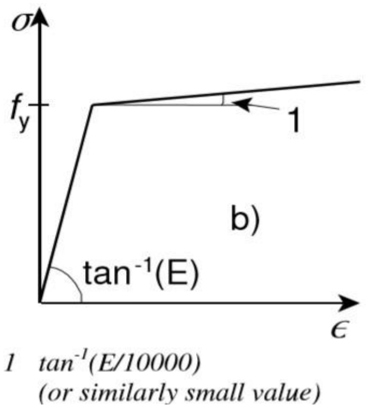


Figure 1.2. Constitutive law of steel plates [5].

1.3 Welds

Welds are one of the connector elements in connections. There are many options for modelling welds.

A widely used option for modelling welds, which is the one used in CYPE Connect, is the direct connection between plates to be welded by means of force-deformation constraints, also known as Multi Point Constraints. The technique for modelling welds using rigid links was suggested by Fayard and Bignonnet (1996) [6] and is based on modelling the local rigidity of the welded connections connecting two adjacent *Shell* elements through their nodes across the entire length of the weld. Using rigid links for modelling welds can be found in many scientific documents such as [7], [8], [9], and [10].

In CYPE Connect, the nodes on the end of the surface to be welded are perpendicularly projected onto the surface to which they are welded. The rigid links connect each one of the nodes at the end of the surface to be welded and the projected nodes. Furthermore, these projected nodes are linked using MPC with the surrounding nodes considering the interpolation functions of the deformation field of the element onto which the node is projected. By applying this method, the throat thickness of the weld is respected as well as the weld's actual configuration.

The main advantage of this method is that it allows meshes of different densities to be connected. In image 1.3, the modelling of the weld using rigid elements and MPC in CYPE Connect can be observed.

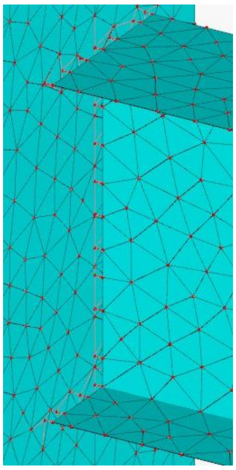


Figure 1.3. Weld modelling in CYPE Connect.

1.4 Bolts

Three different behaviours can be seen in the elements that model bolts. The tensile and shear behaviour of the bolt thread, transmission of tensile forces to the plate, and plate support behaviour in the bolt.

1.4.1 *Thread behaviour*

Non-linear springs simulating tensile behaviour and linear springs simulating shear behaviour will be used to model the behaviour of the bolt thread. The initial rigidities will be obtained from the corresponding standard. For modelling the elastoplastic tensile behaviour of the thread, a bilinear material law is used which is based on stress-strain curve for bolts proposed in different research studies such as [11], where the last stress is produced for a deformation of 5%. The bilinear law of this behaviour can be seen in figure 1.4.

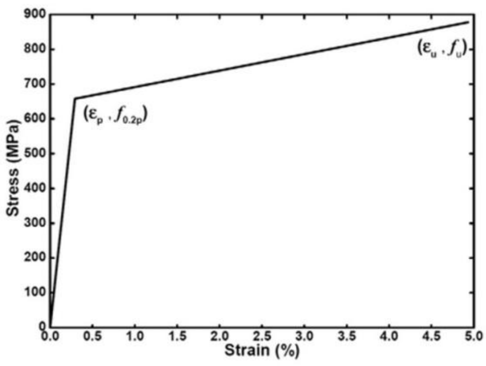


Figure 1.4. Stress-strain of the bolt diagram [11].

This model of the bolt thread using spring-type elements with stiffnesses obtained from standards can also be found in other scientific documents such as section 7.5 in [4].

Below is an example of the equations representing these bilinear laws applying the stiffnesses found in the Eurocode.

Tensile behaviour

The bilinear force-displacement law of tensile behaviour which is based on the behavioural law mentioned above in section 1.4.1 can be seen in figure 1.5.

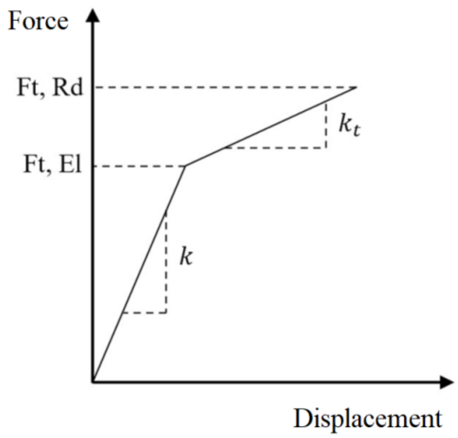


Figure 1.5. Force-displacement of the tensile bolt diagram.

In accordance with EN 1993-1-8 table 6.3.2 [12] the initial k rigidness will be:

$$k = \frac{EA_s}{L_s} \tag{1.1}$$

where E is the modulus of elasticity of the bolt, A_s is the area of the effective area of a stressed cross-section bolt (threaded area) and L_b the stretch length, i.e., the grip length of the bolt (total thickness of the material supported by the bolt) thickness of the washers, half the sum of the height of the nut and half the sum of the height of the bolt head.

The equations describing the bilinear law are:

$$F_{t,Rd} = \frac{k_2 f_{ub} A_s}{\gamma_{M2}} \tag{1.2}$$

$$F_{t,El} = \frac{k_2 f_{yb} A_s}{\gamma_{M2}} \tag{1.3}$$

$$c = \left(\frac{f_{us} - f_{yb}}{0{,}05 - \frac{f_{yb}}{E}} \right) \Big/ E \quad (1.4)$$

$$k_t = c \cdot k \quad (1.5)$$

where:

- $F_{t,Rd}$: The tensile resistance of the bolt according to EN 1993-1-8 table 3.4 [12]
- k_2 = 0.9 (0.63 for countersunk bolt)
- f_{ub}: Ultimate tensile strength of the bolt (ISO 898:2013 [13])
- f_{yb}: Yield tensile strength of the bolt (ISO 898:2013 [13])
- γ_{M2}: Partial safety coefficient EN 1993-1-8 table 2.1 [12] (recommended value γ_{M2} = 1,25)

Shear behaviour

According to the information mentioned in [4], the shear behaviour of a bolt thread can be modelled using a linear spring with a certain stiffness.

According to EN 1993-1-8 table 6.3.2 the stiffness of shear bolt k will be:

$$k = \frac{16 n_b d^2 f_{ub}}{d_{M16}} \quad (1.6)$$

- n_b: Number of shear bolt rows
- d: Diameter of the bolt
- d_{M16}: Diameter of bolt M16, 16 mm

Considering what was shown in [4], the stiffness of a shear bolt can be considered as

$$k = \frac{8 d^2 f_{ub}}{d_{M16}} \quad (1.7)$$

1.4.2 *Transmission of tensile forces to plates*

The meshing of the plate with bolt holes will be carried out as shown in figure 1.6.

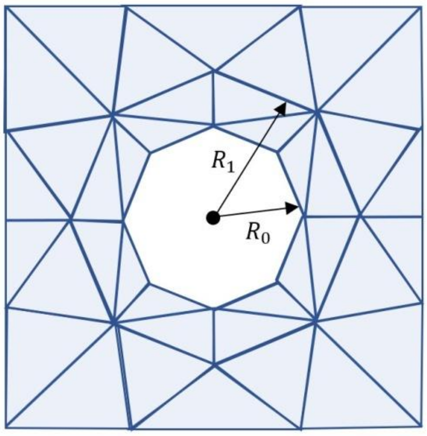


Figure 1.6. Meshing plate with hole.

The hole diameter is $D = 2R_0$ and the R_1 parameter represents the length of the links described below.

The transmission behaviour for tensile forces to the plate is modelled using rigid links between the node from the centre of a hole and the nodes from the outer octagon that transmit tensile forces in the direction perpendicular to the plane of the connected plates

In figure 1.7, the connection between these two interpolation links with nodes from the plate is represented.

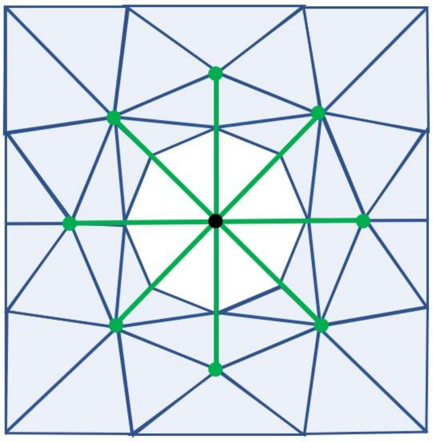


Figure 1.7. Representation of connecting link (green elements) with nodes from the plate.

1.4.3 *Support behaviour*

Support behaviour is modelled using links between the node inside the opening and nodes on the edge of the opening. In these links, the support stiffness of the bolt in the plate is considered. This meshing can be observed in figure 1.8.

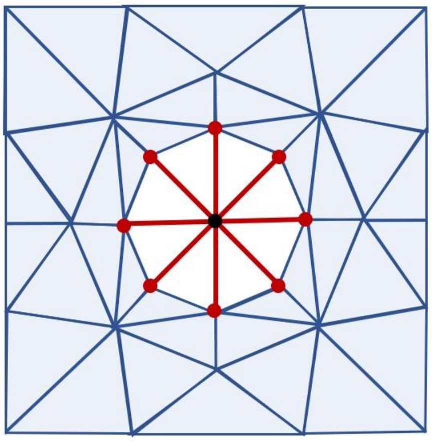


Figure 1.8. Representation of links (red elements) that represent the support behaviour of the plate and bolt.

The global stiffness exerted by these eight connection elements (red elements figure 1.8) is four times the axial stiffness of each of these elements (section 10.3.3 in [4]). This means that whatever the direction of the force applied to the central node common to all eight elements, the apparent stiffness will always be four times the axial stiffness of a radial element.

Considering that these eight elements only work in compression, as is the case for CYPE Connect, the stiffness of each radius, as described in section 10.3.3 in [4], would be:

$$k_N = 0{,}5k \tag{1.8}$$

where k is the support stiffness that is found in different standards.

For the Eurocode, the support stiffness will be as described in EN 1993-1-8 table 6.3.2 [12]

$$k = 24 n_b k_b k_t d f_u \tag{1.9}$$

Considering what was described in section 7.5.1 in [4], the initial support stiffness can be considered as:

$$k = \frac{22{,}5\, t\, d\, f_u}{d_{M16}} \tag{1.10}$$

where:

- f_u: Ultimate tensile stress on the steel where the bolt is supported
- t: Thickness of the component where the bolt is supported
- d: Diameter of the bolt
- d_{M16}: Diameter of bolt M16, 16 mm

1.4.4 *Load transmission*

In CYPE Connect, the bolted connection model is carried out by connecting plates with a bolt model in which the loads are transmitted from the nodes in the first plate to a single node. This node transmits the loads to a second node through an element that simulates the behaviour of the thread, and this second node transmits these loads to nodes on the second plate.

This way of transmitting loads between plates through node-to-node connections can be found in many scientific documents such as in [4], [14], [15], [16], [17] o [18].

1.5 Contact

Contact behaviour between connected components is carried out by including connection elements between nodes that only work in compression with greater stiffness. If the meshing is not compliant, the nodes are projected from one surface to another in a perpendicular direction to the surfaces and the different degrees of freedom using the functions of the shapes of the elements and Mult Point Constraints. The nodes are connected to their corresponding projected nodes and only the elements working in compression are included (figure 1.9) with highly elevated stiffnesses, avoiding proximity between nodes.

Modelling contact relations between node-to-node connections appears in many scientific articles such as in [15], [19] o [20].

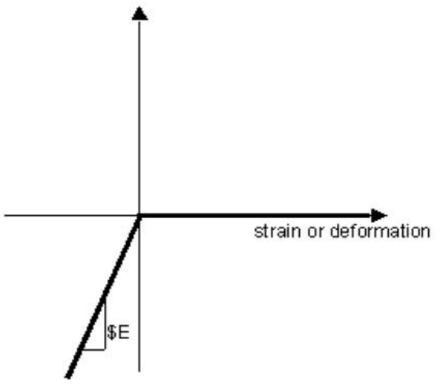


Figure 1.9. Material with compressed behaviour [21].

1.6 Loads and outline conditions

1.6.1 Supports

A member of the pinned connection is always established as "Bearing". All other members are connected to it.

- **Bearing**: The bearing member can be 'continuous' or 'non-continuous' in the connection. The 'non-continuous' members have external fixity on one end and 'continuous' members have external fixity on both ends. External fixities that are introduced coerce displacements and rotations.
- **Connected:** Connected members connect to the bearing element and lack external fixity. Loads are applied to these elements.

1.6.2 Loads

When the structure is analysed by means of a bar model, a detailed model of the connections that make up the structure is not made. If a connection's point is analysed in this type of model, the resulting force in said connection is zero because the model is balanced.

To design the join in a correct way, it is necessary to generate a connecting model that respects the actual layout of the connection, also considering the forces that are applied on each end of the members that make up the connection which are obtained from the resolution of the structure's analysis.

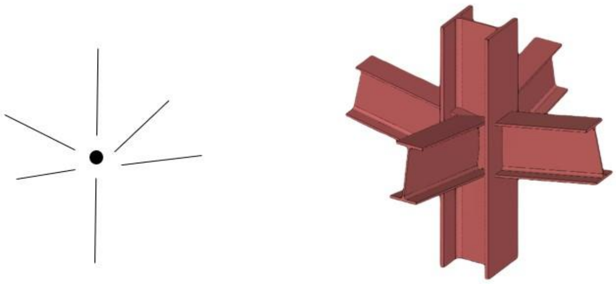
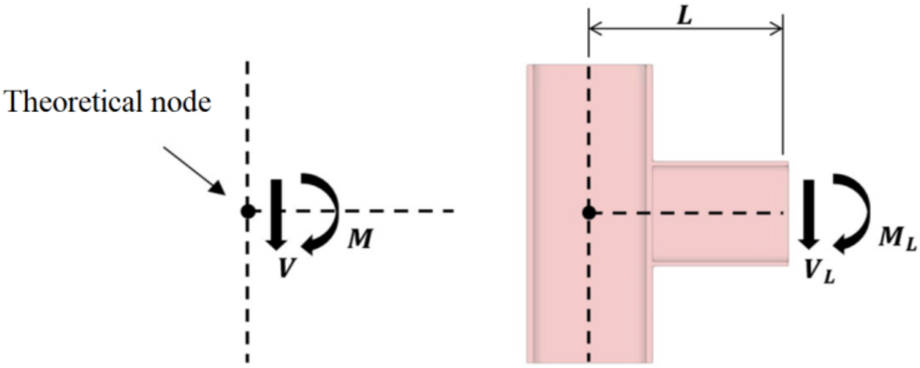


Figure 1.10. Representation of members' theoretical connection and the real connection modelled in CYPE Connect.

The forces obtained in the structural analyses are transmitted to the members' ends. The eccentricities of the members caused by the real design of the connection are respected in this transmission of loads to the ends. In the created connection models, a limited length of the members composing the connection will be considered.

The effects are caused by the precise model of the connection are important for designing the connection. The effects are illustrated in figure 1.11.



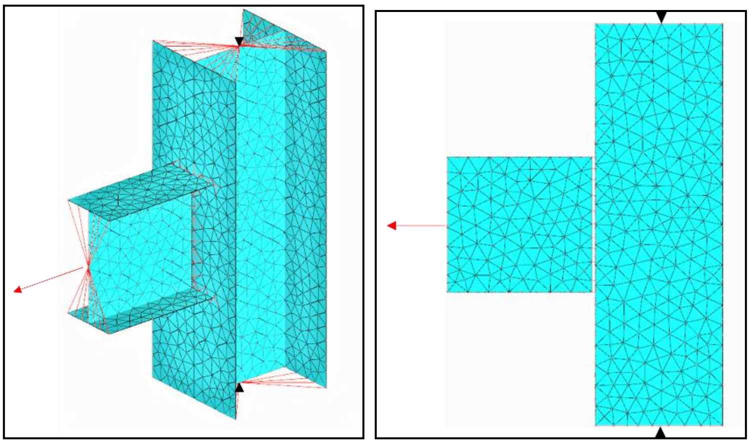
(a) Load application in a theoretical node. (b) Load application at the end of the members.

Figure 1.11. Effects of the connection's precise model in the load transmission to the ends of the members.

When the connection is analysed, it must be considered that the elements and shear forces acting on the theoretical node must be transferred to the ends of the members so that the desired loads are obtained for the theoretical node. In the example shown in figure 1.11, said load transformation would be:

$$V_L = V \qquad M_L = M - V \cdot L \tag{1.11}$$

To model the application of loads at the members' ends, the procedure described in section 10.4.1 in [4] is followed. Fictitious nodes located in the ideal axes of the members will be included and these nodes will be connected to the member end nodes using fictitious rigid elements. An example of this load modelling can be observed in figure 1.12.



(a) General view. (b) Side view.

Figure 1.12. Modelling fictitious rigid elements.

This way of modelling loads has the advantage of reducing local stress concentration, as opposed to the models in which the loads are directly applied to the nodes and the advantage of easily applying the loads and outline conditions onto the different elements. In a way, using this form of modelling considers the Navier or plane section hypothesis in which it is confirmed that plane sections perpendicular to the beam axis before deformation, remain plane and perpendicular to the beam axis after deformation.

2 References

[1] Frank Mckenna, Michael Scott y Gregory Fenves. "Nonlinear Finite-Element Analysis Software Architecture Using Object Composition". In: *Journal of Computing in Civil Engineering J COMPUT CIVIL ENG* 24 (January 2010). DOI: 10.1061 / (ASCE) CP. 1943-5487. 0000002.

[2] Dassault Systemes. "Abaqus/CAE User's Guide." In: (2014).

[3] Xinzheng Lu y col. "Development and Application of a High-Performance Triangular Shell Element and an Explicit Algorithm in Opensees for Strongly Nonlinear Analysis". En: *Computer Modeling in Engineering and Sciences* 120 (January 2019), pages 561-582. DOI: 10.32604/ cmes.2019.04770.

[4] Paolo Rugarli. *Steel Connection Analysis.* John Wiley and Sons, 2018.

[5] "Eurocode 3: Design of steel structures-Part 1-5: General rules-Plated structural elements". In: (2006).

[6] Jean-Luc Fayard y Andre Bignonnet. "Fatigue Design Criterion For Welded Structures". En: *Fatigue and Fracture of Engineering Materials and Structures* 19 (June 1996), pages 723-729.

[7] M. Bennebach, P. Klein y E. Kirchner. "Several seam weld finite element idealizations challenged in fatigue within a French industrial collaborative workgroup". In: *Procedia Engineering* 213 (2018). 7th International Conference on Fatigue Design, Fatigue Design 2017, 29-30 November 2017, Senlis, France, pages 403-417. ISSN: 1877-7058. DOI: https://doi.org/10.1016/ j.proeng.2018.02.040. URL: https://www.sciencedirect.com/science/ article/pii/S1877705818302716.

[8] Jean Abry y col. "Enhanced fatigue structural stress analysis of a heavy vehicle seam welded steel chassis frame: FEA model preparation, weld model description, fatigue stress calculation and correlation with 10 year operating experience". In: *Procedia Engineering* 213 (2018). 7th International Conference on Fatigue Design, Fatigue Design 2017, 29-30 November 2017, Senlis, France, pages 539-548. ISSN: 1877-7058. DOI: https://doi.org/10.1016/j.proeng.2018.02.050. URL: https://www.sciencedirect.com/science/article/ pii/S1877705818302819.

[9] Didier Turlier y col. "Seam weld shell element model for thin walled structure FE fatigue design". In: *MATEC Web of Conferences* 165 (January 2018), page 21007. DOI: 10.1051/matecconf/ 201816521007.

[10] Didier Turlier y col. "New FEA shell element model for seam weld static and fatigue structural assessment". In: *Fatigue design* 165 (November 2011). DOI: 10.1051/matecconf/ 201816521007.

[11] Ying Hu y col. “FE simulation and experimental tests of high-strength structural bolts under tension”. In: *Journal of Constructional Steel Research* 126 (2016), pages 174-186. ISSN: 0143-974X. DOI: https://doi.org/10.1016/j.jcsr.2016.07.021. URL: https://www. sciencedirect.com/science/article/pii/S0143974X16302061.

[12] “Eurocode 3: Design of steel structures-Part 1-8: Design of joints”. : (2005).

[13] ISO 898-1:2013. “Mechanical properties of fasteners made of carbon steel and alloy steel, Part 1: Bolts, Screws and Studs with Specified Property Classes, Coarse Thread and Fine Pitch Thread”. In: *ISO* (2013).

[14] S. Xu y X. Deng. “An evaluation of simplified finite element models for spot-welded joints”. In: *Finite Elements in Analysis and Design* 40.9 (2004), pages. 1175-1194. ISSN: 0168-874X. DOI: https://doi.org/10.1016/j.finel.2003.08.006. URL: https://www. sciencedirect.com/science/article/pii/S0168874X03001598.

[15] Jeong Kim, Joo-Cheol Yoon y Beom-Soo Kang. “Finite element analysis and modeling of structure with bolted joints”. In: Applied Mathematical Modelling 31.5 (2007), pages 895-911. ISSN: 0307-904X. DOI: https://doi.org/10.1016/j.apm.2006.03.020. URL: https://www.sciencedirect.com/science/article/pii/S0307904X0600062X.

[16] Mohamed El Aghoury, Essam Amoush y Mohamed Soliman. “Numerical Study of Bolted Moment Connections in Cold-Formed Steel Frames”. In: *Future Engineering Journal* (2020).

[17] Alexandra Korolija. “FE-modeling of bolted joints in structures”. In: *Master Thesis in Solid Mechanics. Linkopings Universitet*¨ (2012).

[18] Jun Ye y col. “Efficient design of cold-formed steel bolted-moment connections for earthquake resistant frames”. In: *Thin-Walled Structures* 150 (2020). ISSN: 0263-8231. DOI: https:// doi.org/10.1016/j.tws.2018.12.015. URL: https://www.sciencedirect. com/science/article/pii/S026382311830942X.

[19] “ANSYS. Theory Reference Release 5.6.” In: (1999).

[20] A.M. Habraken y S. Cescotto. “Contact between deformable solids: The fully coupled approach”. In: *Mathematical and Computer Modelling* 28.4 (1998). Recent Advances in Contact Mechanics, pages 153-169. ISSN: 0895-7177. DOI: https://doi.org/10.1016/S0895-7177(98) 00115-0. URL: https://www.sciencedirect.com/science/article/pii/ S0895717798001150.

[21] *Elastic-No Tension Material*. https://opensees.berkeley.edu/wiki/index.php/ Elastic-No_Tension_Material. 2010.