NOVEMBRO 25 e 26
DOIS MIL E VINTE UM
XIII Congresso
de Construção
Metálica e Mista

FINITE ELEMENT ANALYSIS OF JOINTS, COMPARISON OF RESULTS WITH THE COMPONENT METHOD ACCORDING TO EN 1993-1-8:2005

Pablo Gómez[a], Amaya Gómez[b] and Paula Assis [c]

[a,b] *Cype Software*
[c] *Top Informatica*

Summary. The current standards are based on the well-known component method for the design of steel structure connections. This method proposes to analyse the joints as a set of more basic components. Its main drawback is the difficulty in extrapolating the method to perform the analysis of complex joints. As an alternative, it is possible to test the joints in the laboratory or to carry out finite element modelling. The purpose of this study is to compare the strength obtained on the basis of the Eurocode 3 criteria with the strength obtained from finite element models of two-dimensional elements in order to validate a model which, a priori, is less accurate than the three-dimensional element model.

1. Introduction

Eurocode EN 1993-1-8:2005 provides an analysis method for estimating the strength of structural connections of steel elements. The joints are modelled as a set of basic components, for which the necessary formulation is given to obtain their strength. The main drawback is the difficulty in extrapolating the method to perform the analysis of joints with a complex geometrical configuration.

Thanks to the use of computer tools and increased computing power, numerous studies and analysis tools have been developed to simulate the behaviour of joints using the finite element method. These studies have mainly focused on carrying out volume finite element models.

As an alternative to these finite element models with volume elements, two-dimensional shell elements can be used. The latter have the advantage of requiring less analysis time and modelling difficulty. The aim of this study is to compare the resistance proposed by the

Eurocode EN 1993-1-8:2005, with the resistance obtained by a shell finite element model. The final objective is to validate the results obtained from a model, a priori, less accurate than the three-dimensional one.

The finite element models of the validation cases have been generated using the CYPE Connect program and analysed with the OpenSees analysis engine [1], which is integrated into this software.

2. Description of the model

In the analysis models generated by CYPE Connect [2] we can identify three main elements: plates, welds and bolts.

2.1 Plates

The flat elements that form the sections and the rest of the plates involved in the joint are discretised by *shell* elements, using the triangular element NLDKGT [3] with three nodes.

These elements take into account membrane behaviour (in-plane tension, compression and shear) and plate behaviour (out-of-plane moment and torsion). In non-linear range, plate behaviour is modelled using layered sections. The plate thickness is divided into a number of layers (in this case 5) where the problem to be solved is plane strain. The analysis of the bending moments is performed by adding the effects of each layer and we can no longer consider that the stresses in the plate thickness are obtained by adding the effects of the membrane and the bending behaviour as would be the case in linear analysis [4].

The constitutive law corresponding to the behaviour of the material for plates and profiles shall be a bilinear function [5].

2.2 Welds

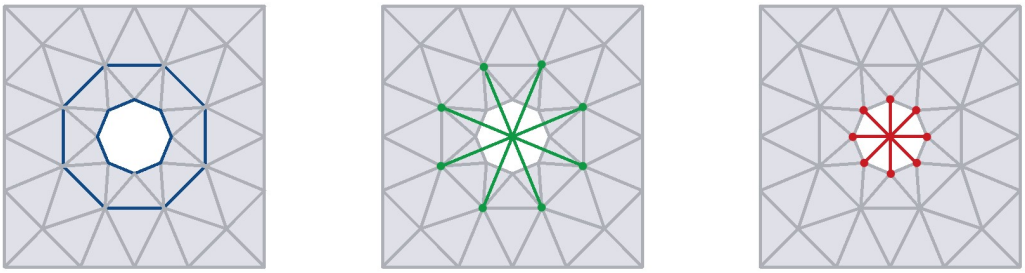
Welds are modelled by direct connection between plates to be welded by means of force-deformation constraints, also known as Multi Point Constraints. The technique of modelling welds by rigid connections was suggested by Fayard and Bignonnet (1996) [6], and can also be found in various scientific papers such as [7], [8], [9] or [10].

2.3 Bolts

To model the elastoplastic behaviour of the shank in tension, a bilinear material behaviour law will be used which is based on bolt stress-strain curves proposed in different research works such as [11], in which the ultimate stress is considered to occur for a strain of 5%. The shear behaviour is modelled according to EN1993-1-8 table 6.3.2 and [4].

2.3.1 Transmission of tensile forces to the plate

The transmission behaviour of tensile forces to the plate is modelled by rigid links between the node at the centre of the opening and the nodes of the outer octagon which transmit tensile forces in the direction perpendicular to the plane of the connected plates, Fig.1.



a) Plate mesh with hole b) Outer octagon links c) Inner octagon links
Fig. 1 Discretisation for bolts

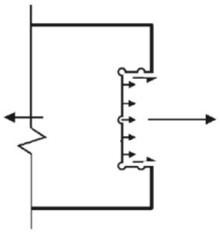
2.3.2 Support
The support effect is modelled by means of links between the inner node of the opening and the nodes of the inner octagon, Fig. 1. These links only work in compression.

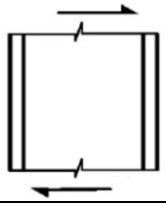
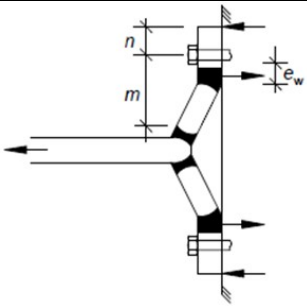
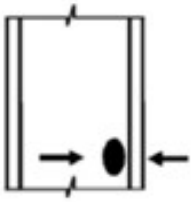
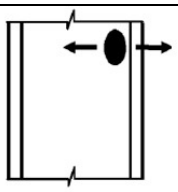
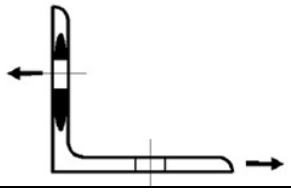
2.3.3 Contact
The contact effect between components that are bolted together is achieved by the inclusion of connecting elements between nodes that work only in compression with very high stiffness. The modelling of contact relationships by means of node-to-node connections is mentioned in numerous scientific papers such as [12], [13] or [14].

3. Comparison method

The process carried out to compare the results has been, firstly, to determine the basic components to be studied, shown in Table 1, and to generate equivalent finite element models in which the most unfavourable resistance is determined by the component being studied. The load equivalent to the resistance calculated by the Eurocode, according to the formulation shown in Table 1, is then introduced into the CYPE Connect model. After the analysis, depending on the equivalent Von Mises deformation obtained, the load is iteratively increased or decreased until an equivalent Von Mises deformation value of 5% is reached (EN 1993-1-5, App. C, Par. C.8, Note 1).
The models studied have been made with version 2021.g of CYPE Connect. The file containing these models can be downloaded from the following link:
http://share2.cype.com/files/Test_EN-1993-1-8_20210825_141923.html

Table 1: Components studied

Component	Resistance	Eq.
Block tearing (EN 1993-1-8:2005, 3.10.2)		
	$V_{eff,1,Rd} = \frac{f_u A_{nt}}{\gamma_{M2}} + \frac{f_y A_{nv}}{\sqrt{3}\,\gamma_{M0}}$	(1)
Column web panel in shear (EN 1993-1-8:2005, 6.2.6.1)		

	$V_{wp,Rd} = \frac{0.9 f_y A_{nv}}{\sqrt{3}\,\gamma_{M0}}$	(2)
Equivalent T-stub in tension (EN 1993-1-8:2005, 6.2.4)		
	$F_{T,1,Rd} = 4 \cdot M_{pl,1,Rd}/m$ $M_{pl,1,Rd} = \frac{0.25\, l_{eff,1}\, t_f^2\, f_y}{\gamma_{M0}}$	(3) (4)
Column web in transverse compression (EN 1993-1-8:2005, 6.2.6.2)		
	$F_{c,wc,Rd} = \frac{\omega\, k_{wc}\, f_y\, t_{wc}\, b_{eff,c,wc}}{\gamma_{M0}}$ $F_{c,wc,Rd} \leq \frac{\omega\, k_{wc}\, \rho\, f_y\, t_{wc}\, b_{eff,c,wc}}{\gamma_{M0}}$	(5) (6)
Column web in transverse tension (EN 1993-1-8:2005, 6.2.6.3)		
	$F_{t,wc,Rd} = \omega\, f_y\, t_{wc}\, b_{eff,c,wc}/\gamma_{M0}$	(7)
Flange cleat in bending (EN 1993-1-8:2005, 6.2.6.6)		
	$F_{T,1,Rd} = \frac{(8n - 2e_w) M_{pl,1,Rd}}{2mn - e_w(m+n)}$	(8)

4. Finite element models

Simplified models have been created, Fig.2, looking for the failure mode to occur for a different component in each case. The type of steel used is S275 in all cases.

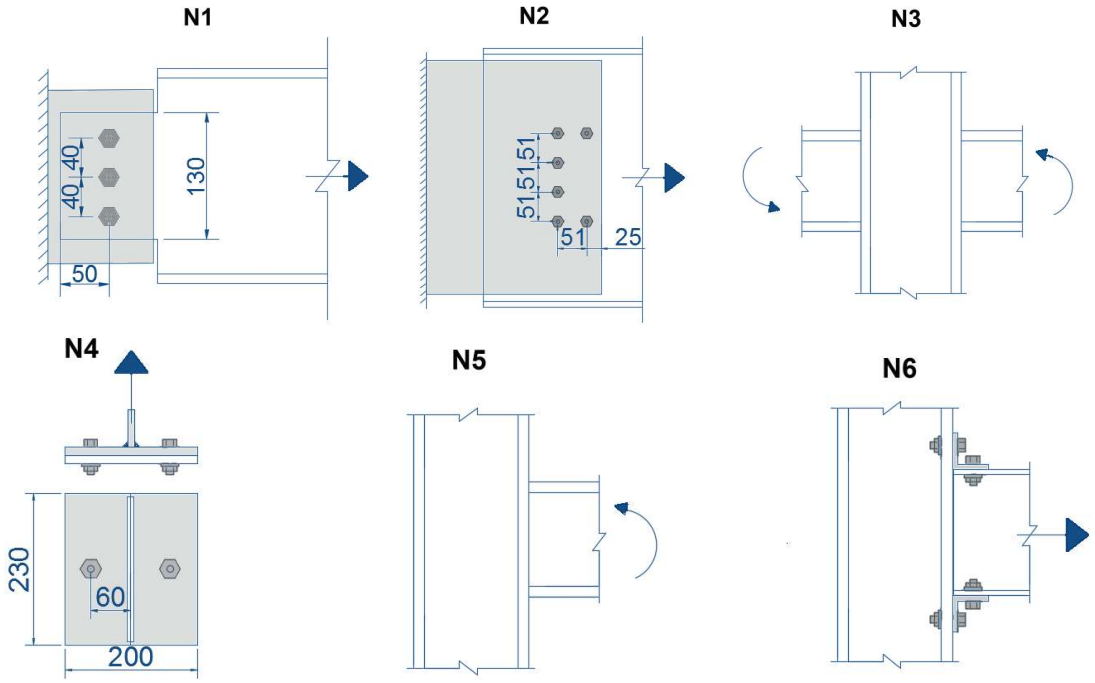


Fig. 2 Sketches of the studied models

4.1 Description

4.1.1 Model N1, block tearing
Beam web bolted to two 9 mm thick side plates so that the most unfavourable component is the web of the beam. The trimmed edge of the web is 130 mm, thickness 6 mm, resulting in a net tensile area equal to 456 mm^2. A tension axial force is introduced into the beam. M16 bolts, quality 8.8.

4.1.2 Model N2, block tearing
The model consists of a 6.6 mm thick plate, to which a tension axial force is applied by means of a group of 6 M18 bolts. Resulting in a net tensile area equal to 613.8 mm^2 and a net shear area equal to 613.8 mm^2.

4.1.3 Model N3, column web panel in shear
The N3-A model consists of a column to which each flange is joined to a beam by welding its flanges. Moments around the y-axis are introduced to each beam equal and opposite sign. All sections are HEA240 cross-section. The N3-B model is similar to the N3-A model, but reduces the width of the column flanges by half, 120 mm.

4.1.4 Model N4, equivalent single T-stub in tension
The failure mode sought for the equivalent T-stub is that of complete yielding of the flange (Mode 1). The model consists of a beam connected to a column flange by a 10 mm thickness end plate, with a row of 2 M18 bolts of quality 8.8. Only the web of the beam is welded. The test focuses on the result of the end plate, therefore the column flange and the beam web are introduced with increased stiffness.

4.1.5 Model N5, column web in transverse compression and tension

Connection between column and beam, the flanges of the beam are welded to the flange of the column, both profiles are of type HEA 240.

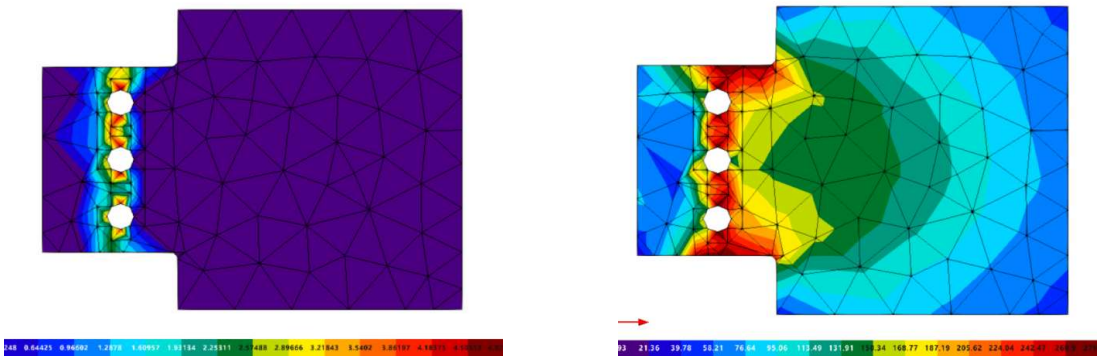
4.1.6 Model N6, flange cleat in bending

The failure mode sought for the equivalent T-stub is that of complete yielding of the flange (Mode 1). The model consists of a beam-column connection, in which the flanges of the beam are joined to the flange of the column by means of two bolted angles. Tension axial force is applied to the beam until a deformation of less than the limit deformation is achieved. The bolts used are of high quality, 10.9, with the aim of ensuring that failure is produced by Mode 1. The diameter of the bolts is defined as 12mm.

4.2 Results

4.2.1 Model N1

A deformation of less than 5% is obtained, Fig.3, for a load value equal to 153.5 kN.

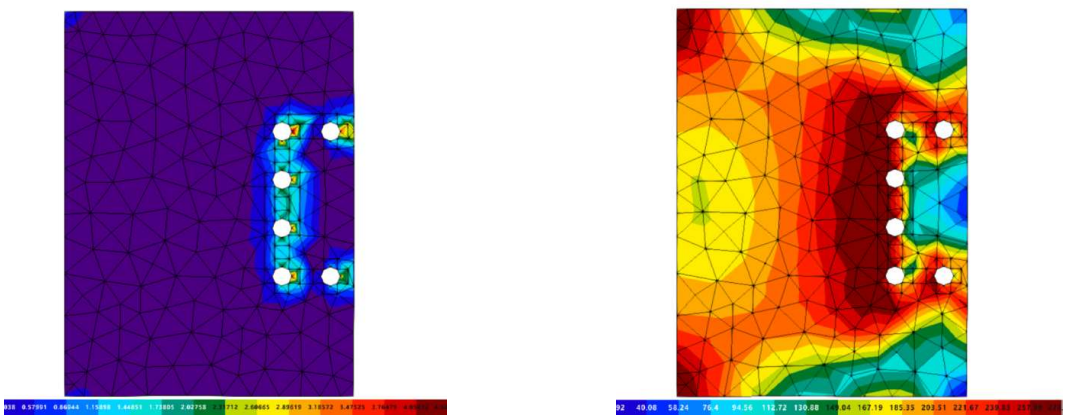


a) Maximum deformation 4.82%. b) Maximum Von Mises stress 279.32 MPa

Fig. 3 Results of model N1

4.2.2 Model N2

A deformation of 4.34% is obtained, Fig.4, for a load value equal to 352 kN. In this case, the obtained resistance is higher than that proposed by the Eurocode but lower than that estimated according to the formulation proposed by [15].



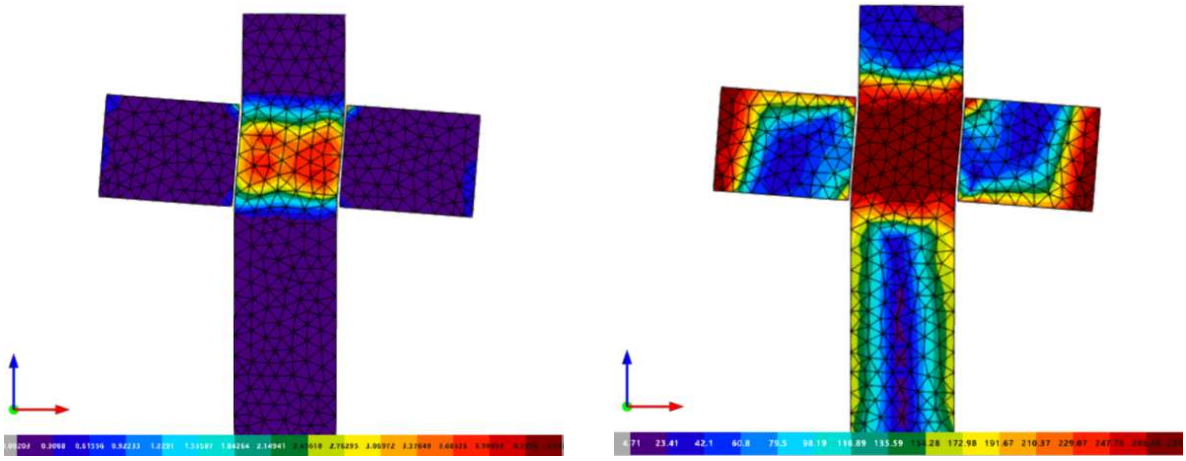
a) Maximum deformation 4.34%. b) Von Mises stress 276.15 MPa

Fig. 4 Results of N2 model

4.2.3 Model N3

In the finite element model A, a deformation of 4.50 % is obtained, Fig. 5, for a bending moment value in each beam of 41.5 kNm, which would produce a shear force of 377.27 kN.

In the finite element model B, a deformation of 4.08% is obtained for a bending moment value in each beam of 37 kNm, which would produce a shear force of 336.36 kN.



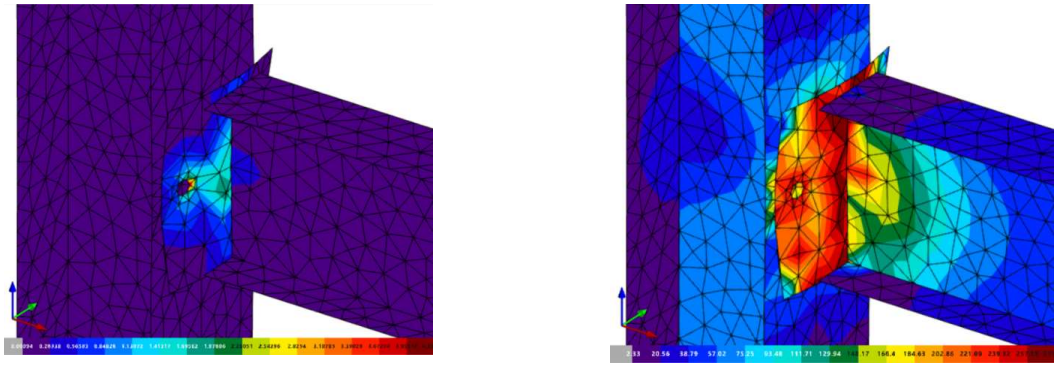
(a) Maximum deformation (b) Von Mises stress

Fig. 5 Results of model N3 B

The comparison shows that the strength value obtained in CYPE Connect approaches the strength proposed by the Eurocode as the width of its flanges is reduced, and therefore the contribution to the strength is lower.

4.2.4 Model N4

Deformation of less than 5% is obtained, Fig.6, for a load value equal to 124 kN.

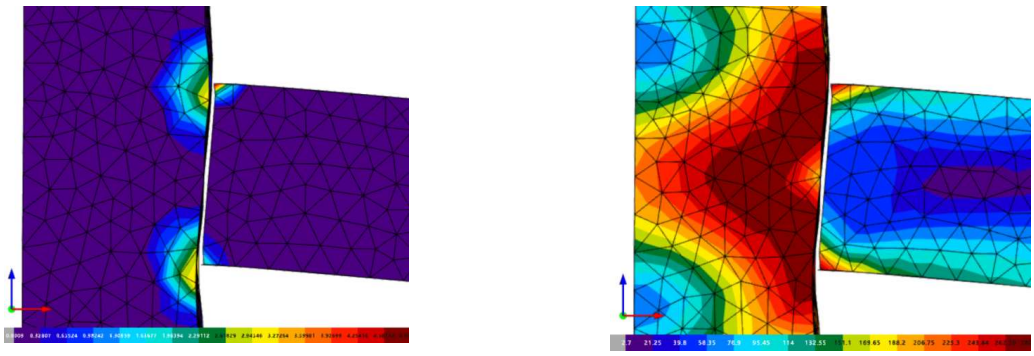


a) Maximum deformation 4.23%. b) Von Mises stress 275.78 MPa

Fig. 6 Results of N5 model

4.2.5 Model N5

A 76 kNm bending moment is applied to the beam in the model. This force produces a deformation of 4.90%, Fig.7. The compressive stress would occur at the bottom flange of the beam, with an estimated value of $(76\ kNm/0.218m) = 348\ kN$. The tensile stress would occur in the upper flange with an estimated value of the same magnitude.



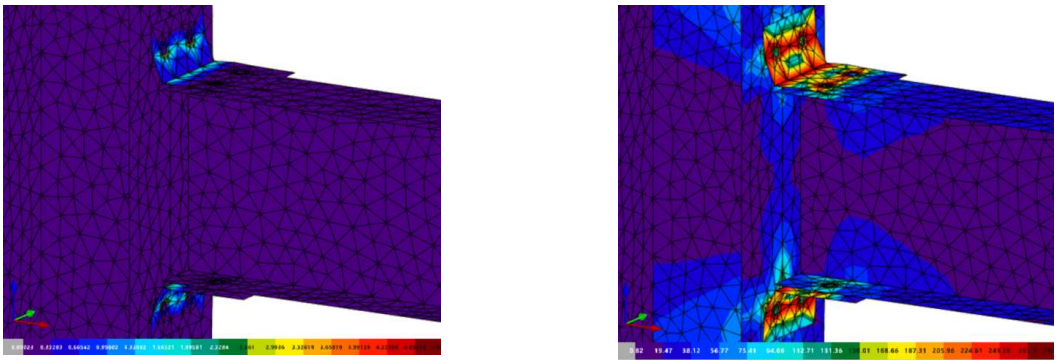
a) Maximum deformation 4.90%. b) Von Mises stress 262.39 MPa

Fig. 7 Results of N5 model

A similar strength value is observed for compression and tension failure without taking into account the plate buckling effect. For the case studied, where no plate buckling check has been performed, a strength variation of 6% is observed.

4.2.6 Model N6

A deformation of 4.98% is obtained, Fig.8, for a load value equal to 101.8 kN (50.9 kN per angle).



a) Maximum deformation 4.98%. b) Von Mises stress 280.5 MPa

Fig. 8 Results of model N6

5. Analysis of results

From the results obtained, shown in Table 2, a great similarity in the strength obtained by both methods is observed, with an average difference of 2.88 %. The largest difference is obtained in the N2 (block tearing) model, with the Eurocode offering a conservative strength, as also indicated in [15]. Looking at the results of models N1 and N2, the main difference can be seen for the tearing in shear zones.

In the N3 model, where the shear resistance of the column web is studied, a difference of 11.98% is observed in model A, which is reduced to -0.16% in model B. The difference between the two models occurs because the formulation proposed by EN 1993-1-8:2005 does not take into account the influence of the column flanges, thus giving a conservative result as the flange size increases.

From model N5 (compression), a difference of 6.09% is obtained due to not taking into account the reduction in resistance due to plate buckling in the finite element model.

Table 2: Summary of results

Model	**EN 1993-1-8 (a)**		**MEF CYPE Connect Steel (b)**	**% difference** $\left(\frac{100 \cdot b}{a}\right) - 100$
	Formulation	Resistance		
N1	Eq. (1)	156.86 kN	153.5 kN	-2.14
N2	Eq. (1)	303.96 kN	352 kN	15.80
N3 A	Eq. (2)	336.9 kN	377.27 kN	11.98
N3 B	Eq. (2)	336.9 kN	336.36 kN	-0.16
N4	Eq. (3)	128.57 kN	124 kN	-3.55
N5 Compression	Eq. (5)	328.03 kN	348 kN	6.09
N5 Traction	Eq. (7)	347.49 kN	348 kN	0.15
N6	Eq. (8)	52.79 kN	50.09	-5.11
			Average difference	**2.88**

6. Conclusions

The finite element model of two-dimensional shells used in CYPE Connect gives similar results to those expected according to EN 1993-1-8, as can be verified in the case studies. For the design of joints with complex geometries, the component method is not easily extrapolated. The proposed finite element analysis method, on the other hand, can be applied without any additional difficulty to simulate the behaviour of any type of joint. The conclusion of this work, in the light of the above, is that the analysis method used in CYPE Connect software is an excellent alternative to the traditional analysis according to the component method.

Notation

A_{nt} Net area subjected to tension
A_{nv} Net area subjected to shear
$F_{c,wc,Rd}$ Design resistance of an unstiffened column web in compression
$F_{t,wc,Rd}$ Design resistance of an unstiffened column web in tension
$F_{T,1,Rd}$ Design resistance of the T-stub flange for mode 1
$M_{pl,1,Rd}$ Design plastic moment resistance for mode 1
$V_{eff,Rd}$ Design block tearing resistance
$V_{wp,Rd}$ Design plastic shear resistance of an unstiffened column web panel
$b_{eff,c,wc}$ Effective width of column web in compression
e_w 0.25 times the diameter of the washer
f_u Ultimate tensile strength of the steel
f_y Yield strength of the steel
k_{wc} Reduction factor
$l_{eff,1}$ Effective length for a non-circular pattern
m Distance from the bolt axis to the root of the weld or radius
n Distance from the bolt axis to the edge of the plate, not greater than 1.25·m
t_f Flange thickness
t_{wc} Thickness of the Column web
γ_{M2} Partial safety factor for the ultimate strength of cross-sections in tension

γ_{M0} Partial safety factor for cross-section resistance
ρ Reduction factor for plate buckling
ω Reduction factor to allow for the possible effects of interaction with shear in the column web panel

References

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

[2] "CYPE Connect Steel - CALCULATION MEMORY." [Online] Available at: http://www.cype.net/manuales/cype_connect_steel/cype_connect_steel_mem_es.pdf. Accessed on: Feb., 8, 2021

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

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

[5] "Eurocode 3: Design of steel structures-Part 1-5", 2006.

[6] Fayard J-L, Bignonnet A. "Fatigue Design Criterion For Welded Structures". *In: Fatigue and Fracture of Engineering Materials and Structures 19*, 723-729, 1996.

[7] Bennebach M, Klein P, Kirchner E. "Several seam weld finite element idealizations challenged in fatigue within a French industrial collaborative workgroup". *In: Procedia Engineering 213,* 2018.

[8] Abry J et al. "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*, Senlis, France, 539-548, 2017. ISSN: 1877-7058.

[9] Turlier D et al. "Seam weld shell element model for thin walled structure FE fatigue design". *In: MATEC Web of Conferences 165*, 21007/2018. DOI: 10.1051/matecconf/ 201816521007.

[10] Turlier D et al. "New FEA shell element model for seam weld static and fatigue structural assessment". *In: Fatigue design 165,* 2011.

[11] Ying Hu et al. "FE simulation and experimental tests of high-strength structural bolts under tension". *In: Journal of Constructional Steel Research 126*, 174-186, 2016. ISSN: 0143-974X.

[12] Kim J, Yoon J-C, Kang B-S. "Finite element analysis and modeling of structure with bolted joints". *In: Applied Mathematical Modelling 31.5*, 895-911, 2007. ISSN: 0307-904X.

[13] ANSYS. Theory Reference Release 5.6. In: (1999).

[14] Habraken A.M. Cescotto S. "Contact between deformable solids: The fully coupled approach". *In: Mathematical and Computer Modelling 28.4. Recent Advances in Contact Mechanics*, 153-169, 1998. ISSN: 0895-7177. DOI: https://doi.org/10.1016/S0895-7177(98) 00115-0.

[15] Topkaya C. "A finite element parametric study on block shear failure of steel tension members", *Journal of Constructional Steel Research*, Ankara, Ankara, 2004.