FINITE ELEMENT ANALYSIS OF T-JOINTS OF TUBULAR STRUCTURES WITH SHS AND RHS SHAPES

Pablo Pajarón

Industrial Engineer
Universidad Politécnica de Cartagena
C/Doctor Fleming, 30202 Cartagena (Murcia), Spain
e-mail: p.pajaron@hotmail.es, web page: http://www.upct.es/

International Center for Numerical Methods in Engineering (CIMNE)
Universidad Politécnica de Catalunya
Campus Norte UPC, 08034 Barcelona, Spain
e-mail: p.pajaron@hotmail.es, web page: http://www.cimne.com/

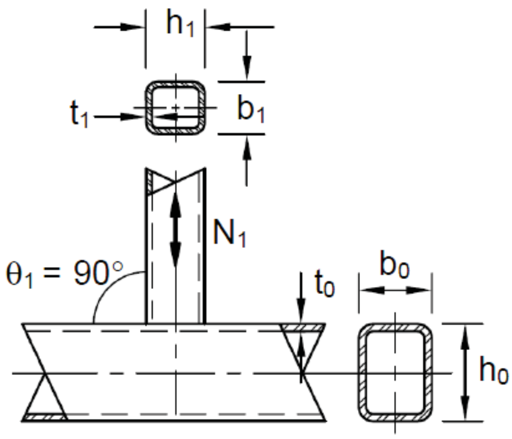
Key Words. Metal structures; Tubular joints; Numerical methods; Structural analysis by finite element method; Non-linear analysis his paper describes the numerical investigation of cold-formed stainless steel tubular T-joints, X-joints and X-joints with chord

Abstract. This paper describes the numerical investigation of cold-formed stainless steel tubular T-joints, using finite element analysis. The stainless steel joints were fabricated from square hollow section (SHS) and rectangular hollow section (RHS) brace and chord members. The geometric and material nonlinearities of stainless steel tubular joints were carefully incorporated in the finite element models. The joint strengths, failure modes as well as load-deformation curves of stainless steel tubular joints were obtained from the numerical analysis. The nonlinear finite element models were calibrated against experimental results of cold-formed stainless steel SHS and RHS tubular T-joints. Good agreement between the experimental and finite element analysis results was achieved. Therefore, a Deformation Limit Criteria was carried out using the verified finite element models to evaluate the effects of the strength and behavior of cold-formed stainless steel tubular joints. The joint strengths obtained from the Deformation Limit criteria in models and the tests were compared with the current design strengths calculated using the European Standard for stainless steel structures, CIDECT and Eurocode design rules for carbon steel tubular structures. It is shown that the design strengths calculated using the European Standards are generally more conservative than those calculated using the Deformation Limit Criteria in models.

1 INTRODUCTION

The use of structures with tubular profiles has increased significantly in recent years, mainly due to the benefits associated with the structural behavior of these profiles and the great aesthetic finish presenting in these structures. Improved manufacturing processes and the development of tubular structures in European countries, Southeast Asia, North America, Australia and Canada, motivated the creation of standards that regulate the behavior and design of joints with these profiles. Furthermore, the development of finite element software has become a powerful tool for simulating the behavior of any structure. Thus it can provide an infinite number of results simulating tubular profiles joints quickly and inexpensively. With the validation of a numerical model based on a single experimental test, it can be simulated an immense amount of real tests varying the dimensions and materials of the joint.

In this document a literature review is performed to obtain experimental tests on tubular T-joints. Numerical models of stainless steel hollow sections T-joints are developed in ANSYS v.14.5 program. The models are validated with the experimental test responses and by using sundry factors (such as welding or the bottom plate lifting when the joint is compressed) to obtain a correct simulation of the reality. The main parameters and ratios of this joint can be seen in Fig.1.



Ratios:

$$\beta = \frac{b_1}{b_0} \tag{1}$$

$$\eta = \frac{h_1}{b_0} \tag{2}$$

$$2\gamma = \frac{b_0}{t_0} \tag{3}$$

$$\tau = \frac{t_1}{b_0} \tag{4}$$

Figure 1. Main parameters and ratios

Failure modes are obtained by applying a compressive force onto the T-joint and the maximum allowable forces (before a failure occurs) are determined by a Deformation Limit Criteria. The results of the models are compared with those that occurred in the experimental tests and the results proposed by CIDECT and Eurocode 3 formulations. Finally it is analyzed whether the proposed formulations in these European standards allow the use of tube thicknesses less than 2.5 mm. and the use of stainless steel in the T-joint´s members. Assuming this, there are significant cost savings in the development of small tubular structures.

2 LITERATURE REVIEW

2.1 Experimental test

R. Feng and B. Young developed several experimental tests based in rectangular hollow section (RHS) and square hollow section (SHS) T-joints 2008 [3], subjected to compression, where the force-displacement curves of the joints were determined as well as different failure modes depending on the value of β ratio. In 2011 [4], they performed numerical models using finite elements software ABAQUS. Where loading and boundary conditions, materials and welding were established, and failure modes and force-displacement joint curves were validated with their experimental tests from previous years and the regulations of EC3 [1] and CIDECT [2]. These experimental tests were used in this document to create two different models with plate thickness less than 2 mm.

The geometry and disposition of the hollow section T- joint, as well as the welds can be seen in Fig.2. Notice that there are to different welds. In one hand there is a weld when the width of the brace is less than the width of the chord β<1, by the other hand there is a full weld when the width of the brace is equal to the width of the chord β=1.

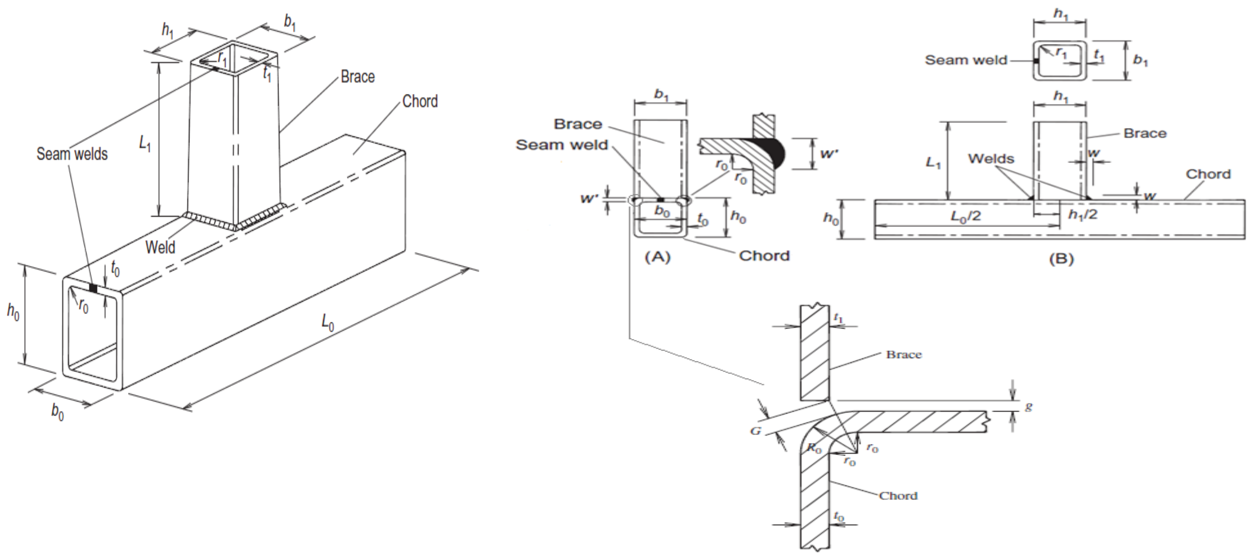


Figure 2. Geometry and disposition of hollow section profiles

The schematic sketches of the testing bench are shown in Fig.3a for the end view and elevation, respectively. Axial compression force was applied to the specimen by using a servo-controlled hydraulic machine. The upper end support was movable to allow tests to be conducted at various specimen dimensions. A special fixed-ended bearing was used at the end of the brace member, so that a uniform axial compression load can be applied to the specimen. The special bearing was connected to the upper end support. The chord member of the test specimen was rest on the bottom end plate, which connected to the bottom support of the testing machine. This provided support to the entire chord member.

Two displacement transducers were positioned on either side of the brace member measuring the vertical deflections at the center of the connecting face of the chord. The transducers were positioned 20mm away from the faces of the brace member, as shown in the Fig.3b

a) b)

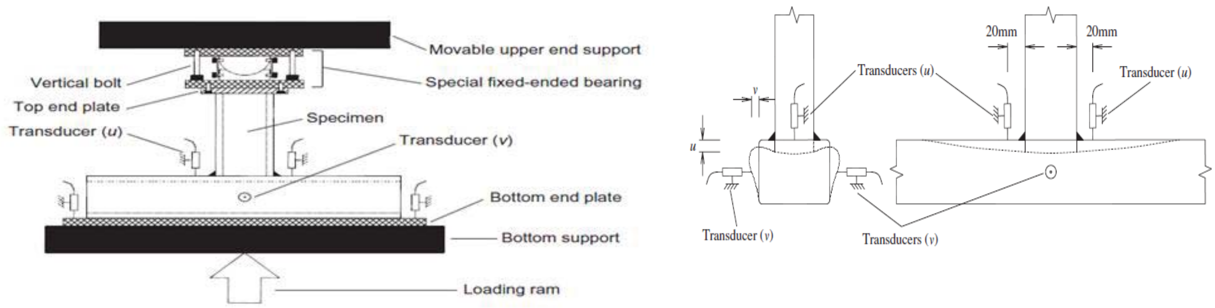


Figure 3. Testing bench and transducers

2.2 Deformation Limit Criteria

During the last decades many researchers have sought a deformation limit criteria when the members are subjected to axial force in order to assure the correct behavior and safety of the structure. In 2000 Zhao [5] proposed a Deformation Limit Criteria since there is not a clear maximum plastification point when a joint is subjected to compression.

Case 1: $\beta \leq 0{,}8$

- If $N_{3\%b0}$ (corresponding to a strain equal $3\%b_0$) is less than $1{,}5N_{1\%b0}$ (corresponding to a strain equal $1\%b_0$), then failure force is $N_f = N_{3\%b0}$. (Fig. 4a).
- If $N_{3\%b0}$ is upper than $1{,}5\,N_{1\%b0}$, then failure force is $N_f = 1{,}5N_{1\%b0}$. (Fig. 4b).

a) b)

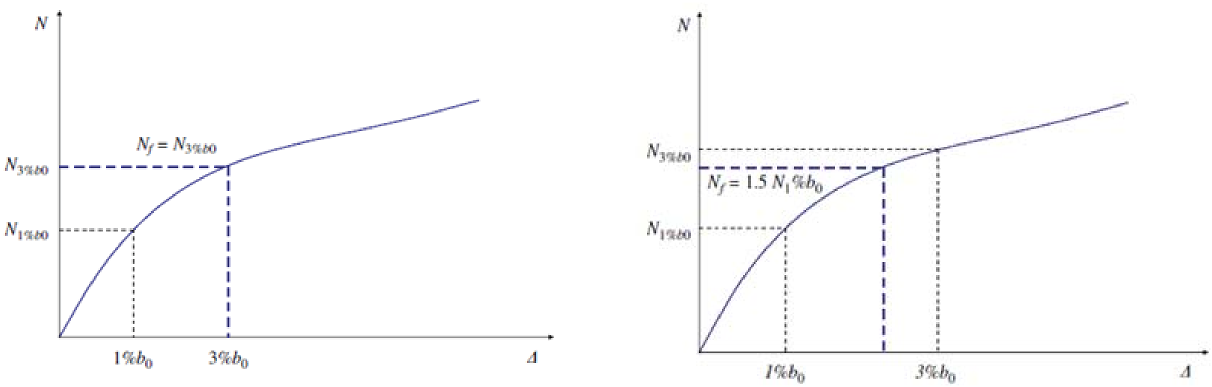


Figure 4. Force-deformation curves

Case 2: 0,8<$\beta \leq$ 1,0

- Failure force is equal to the maximum experimental test force (N_f=$N_{máx}$), if the strain associated to this force is less than 3%b_0. (Fig. 5c).
- If the maximum experimental test force is upper than the force associated a strain equal to 3%b_0, then the failure force is N_f=$N_{3\%b0}$. (Fig. 5d).

c) d)

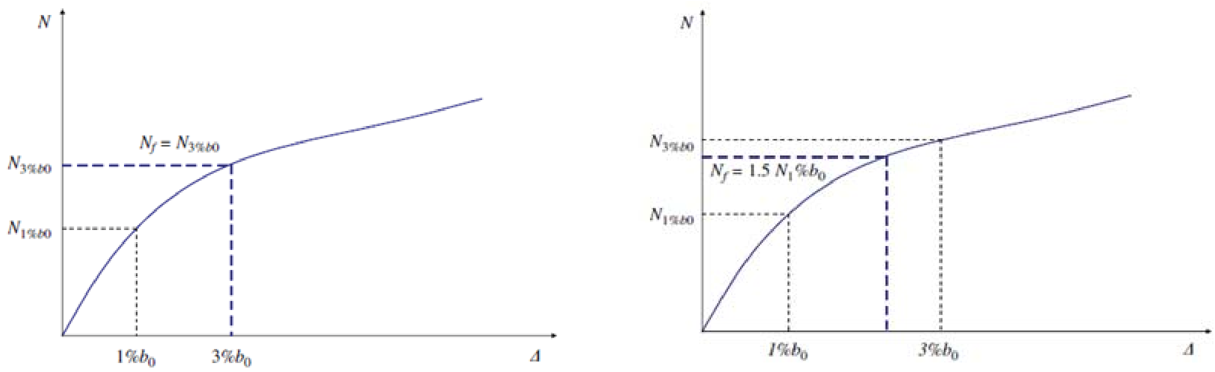


Figure 5. Force-deformation curves

3 FINITE ELEMENT MODELS

3.1 General

The general purpose finite element program ANSYS was used for the numerical modelling of cold-formed stainless steel tubular T-joints. Two FE models were developed, namely Model 1 and Model 2. The load–displacement nonlinear analysis was performed by setting it in ANSYS. Both geometric and material nonlinearities have been taken into account in the finite element models. In order to provide accurate results with a reasonable computational cost, the element type and mesh size of the stainless steel tube and the welding material were carefully determined by convergence studies for the simulation of cold-formed stainless steel tubular joints. The joint strengths, failure modes as well as load–deformation curves were obtained from the numerical analysis. The finite element analysis includes various important factors, such as the modelling of materials and welds, contact interaction between the T-joint specimens and the supporting plate, and loading and boundary conditions. The dimensions of the test specimens are detailed in Feng and Young [3] for cold-formed stainless steel tubular T-joints.

Solid element 30 nodes of 3 freedom degrees per node, SOLID186 was used, this element is available in ANSYS software per default. This element has the ability to develop a mixed formulation for simulating deformations of materials elastoplastic almost incompressible and compressible hyperelastic materials. The use of solid elements rather than shell elements for modeling of the joints could achieve accurate results with a slight increase of computational time.

3.2 Models

The members of the T-joint were extruded from rectangular sketches and then joined by the welds. To facilitate the convergence several cuts were made in the joint the a quarter of tubular joint was modelled by taking advantage of two planes of symmetry in geometry, loading application and boundary conditions. The nodes on symmetry surfaces 1 and 2 were restrained against displacements in X and Y directions due to symmetry, respectively, as shown in Fig. 6.

A plate of solid elements was established at the bottom of the joints to allow the lifting of the upper chord plate. The Models 1 & 2 can be seen in Fig. 6.

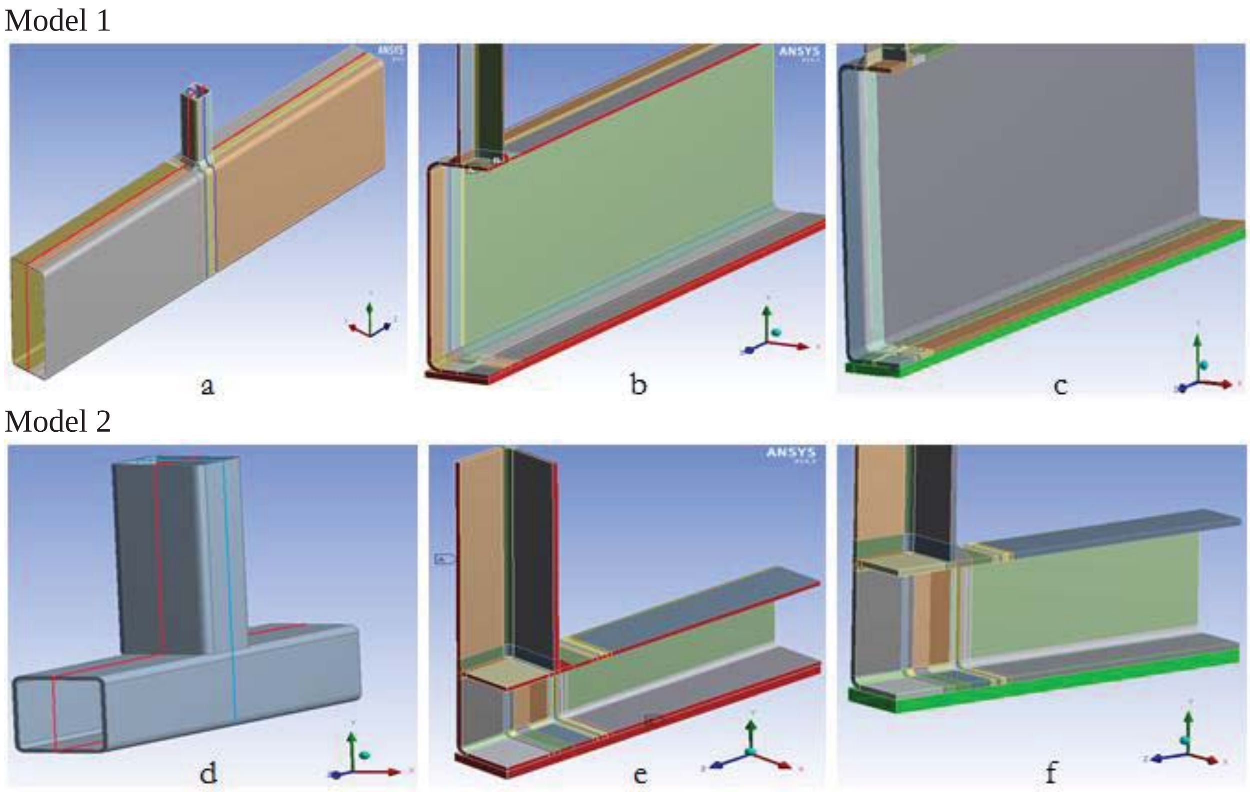


Figure 6. Models

3.3 Mesh

In order to obtain the optimum finite element mesh size, convergence studies were carried out. It was found that the mesh size of approximately 2 mm × 2 mm (length by width) for small specimens, and 12mm×12mm for large specimens in modelling the flat portions of both flange and web elements could achieve accurate results with minimum computational time. In order to facilitate the convergence tetrahedral shapes were used, two layers in the thickness of the plates were applied to improve the model behavior. The mesh of these two model can be seen if Fig. 7.

Model 1 Model 2

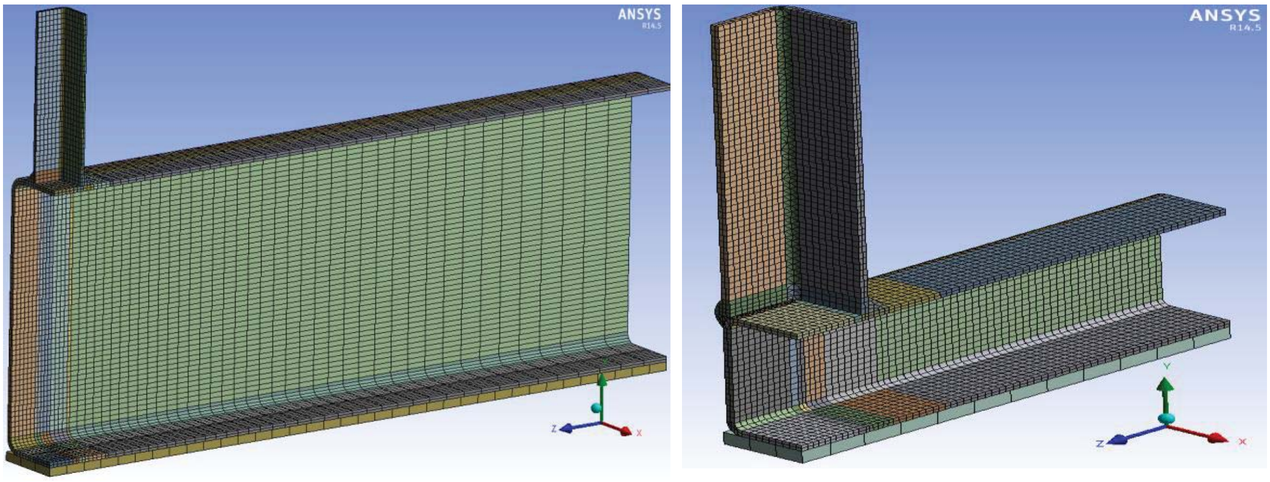


Figure 7. Mesh

3.4 Material modeling

The measured stress–strain curves of stainless steel tubes were used in the finite element models. The static stress–strain curves were determined using the static loads near the proportional limit stress and the ultimate tensile stress. The initial part of the nonlinear stress–strain curve represents the elastic property up to the proportional limit stress with measured Young's modulus (E), and Poisson's ratio equals to 0.3. The nonlinear finite element analysis involved large plastic strains, therefore, the static stress–strain curves obtained from tensile coupon tests were converted to true stress–logarithmic plastic strain curves. The material nonlinearity behavior was included in the finite element models. The true stress and logarithmic plastic strain were calculated based on the recommendation of ANSYS.

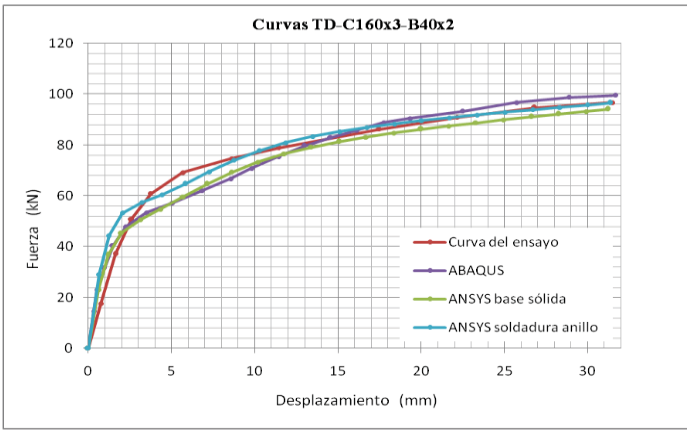
3.5 Loading and boundary conditions

For the modelling of welded tubular T-joints, the top end of brace and the loaded end at the bottom supporting plate were restrained against all degrees of freedom, except for the displacement at the loaded end in the direction of the applied load. The nodes other than the loaded end and the top end were free to translate and rotate in any direction. The load was applied in increments by using steps available in ANSYS. The static uniform loads were applied by means of displacement at each node of the loaded member. The nonlinear geometry parameter was used for the consideration of the large displacement analysis. The contact between the bottom supporting plate and the bottom surface of chord member plays an important role in the load transferring mechanism. Therefore, an analytic rigid-deformable contact interaction between the rigid bottom supporting plate and the deformable chord member was established using an algorithm available in ANSYS. The contact interaction allows the surfaces to separate under the effect of tensile force. However, they are not allowed to penetrate each other under the effect of compressive.

4 MODEL VERIFICATION

A comparison between the experimental and numerical results was carried out to verify the finite element models. The measured dimensions and material properties of the joints were modelled. The joint strengths, load–axial shortening curves and deformed shapes based on the different failure modes of the tubular T-joints have been investigated. The comparison of the joint strengths vs. axial shortening obtained from the tests and finite element analysis are shown in Fig. 8 y Fig. 9. Good agreement between the experimental and finite element analysis results was achieved. A better response in ANSYS models was obtained that one provides for ABAQUS model.

Model 1



Model 2

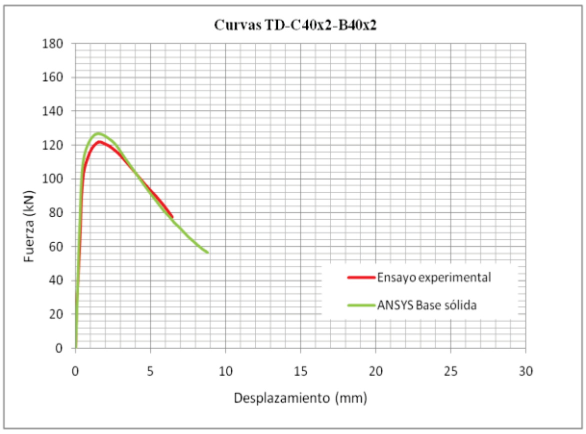


Figure 8. Mesh

5 CONCLUSIONS

– It shows that the finite element simulation is a useful tool for predicting the behavior of T-joints with tubular steel profiles.

– The ring weld proposal improves the model response of this, and the inclusion of rigid elements plate in the base of the T-joint, allows the lifting of the bottom chord plate, providing a realistic simulation.

– The failure modes presented in numerical models correspond to those that occurred in experimental trials and proposed by the standard (EC3 and CIDECT) based in β ratio.

– Deformation Limit Criteria gives an accurate force before Nf before the failure occurs in T joint.

  • European standards provide more conservative resistances than the values obtained by finite elements model.

- Can be used at joints with plate thicknesses of 2 mm, even if they were designed to thicknesses greater than 2.5 mm.

– This would be significant cost savings in the development of small tubular structures with RHS profiles.

6 REFERENCES

[1] UNE-EN 1993-1-8:2011 *Eurocódigo 3: Diseño de estructuras de acero. Parte 1-8: Diseño de uniones.*

[2] J.A. Packer, J. Wardenier, X.L. Zhao, G.J. Van der Vegte, Y. Kurobane. *Design Guide for Rectangular Hollow Section (RHS) Joints Under Predominantly Static Loading* - 2nd Edition, CIDECT, 2009.

[9] Ran Feng, Ben Young. Experimental investigation of cold-formed stainless steel tubular T-joints. *Thin Walled Structures*, 46, pp. 1183-9, 2008.

[4] Ran Feng, Ben Young. Design of cold-formed stainless steel tubular T- and X-joints. *Journal of Constructional Steel Research*, 67, pp. 421-436, 2011.

[5] X-L. Zhao. Deformation limit and ultimate strength of welded T-joints in cold-formed RHS sections, *Journal of Constructional Steel Research*, vol. 53, pp. 149-165, 2000.