COMPUTATIONAL MODELING OF JOINT LINES IN PUNCHED METAL PL

webofscience.com

By: Paiva, L (Paiva, Lucas) [1] ; Braz-Cesar, M (Braz-Cesar, Manuel) [1] ; Paula, A (Paula, Antonio) [1]
Edited by: Gomes, JFS (Gomes, JFS) ; Meguid, SA (Meguid, SA)

View Web of Science ResearcherID and ORCID (provided by Clarivate)

7TH INTERNATIONAL CONFERENCE INTEGRITY-RELIABILITY-FAILURE (IRF2020)
Page: 937-942
Published: 2020
Indexed: 2020-01-01
Document Type: Proceedings Paper

Conference

Meeting: 7th International Conference on Integrity-Reliability-Failure (IRF)
Location: Funchal, PORTUGAL
Date: SEP 06-10, 2020
Sponsors: Univ Porto, Fac Engn; University of Toronto, MADL; Portuguese Soc Expt Mech, APAET; European Soc Expt Mech; Amer Soc Expt Mech; Japanese Soc Mech Engn; Int Measurement Confederat; European Assoc Dynam Mat; Inst Ciencia & Inovacao Engn Mecanica & Engn Ind; Fundacao Ciencia & Tecnologia; ABREU

Abstract

Punched metal plate fasteners (nail plates) are currently the top choice in the wooden-truss industry because of its high mechanical capacity and easy automated assembly. Here, a computational modelling is proposed to evaluate the force distribution within the plate's anchorage area and in the rupture lines, a structural system that in most cases is highly hyperestatic and does not feature an analytical solution. This method can be applied to areas with multiple gaps (rupture lines) and therefore its analysis is discussed from a quantitative and qualitive point of view.

Keywords

Author Keywords: punched metal plate fasteners; wood trusses; numerical analysis

Author Information

Corresponding Address: Braz-Cesar, Manuel (corresponding author)

Polytech Inst Braganca, Appl Mech Dept, Braganca, Portugal

Addresses:
[1] Polytech Inst Braganca, Appl Mech Dept, Braganca, Portugal

E-mail Addresses: brazcesar@ipb.pt

Categories/Classification

Research Areas: Materials Science; Mechanics

Proceedings IRF2020: 7th International Conference Integrity-Reliability-Failure.
J.F. Silva Gomes and S.A. Meguid (editors), INEGI-FEUP (2020), pp.937-942.
https://paginas.fe.up.pt/~irf/Proceedings_IRF2020/

PAPER REF: 17808

COMPUTATIONAL MODELING OF JOINT LINES IN PUNCHED METAL PLATE FASTENERS

Lucas Paiva[1], Manuel Braz-Cesar[2(*)], António Paula[2]
Applied Mechanics Department, Polytechnic Institute of Bragança,
Bragança, Portugal
[(*)]*Email:* brazcesar@ipb.pt

ABSTRACT

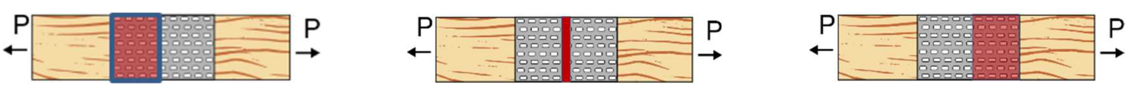
Punched metal plate fasteners (nail plates) are currently the top choice in the wooden-truss industry because of its high mechanical capacity and easy automated assembly. Here, a computational modelling is proposed to evaluate the force distribution within the plate's anchorage area and in the rupture lines, a structural system that in most cases is highly hyperestatic and does not feature an analytical solution. This method can be applied to areas with multiple gaps (rupture lines) and therefore its analysis is discussed from a quantitative and qualitive point of view.

Keywords: punched metal plate fasteners, wood trusses, numerical analysis.

INTRODUCTION

Metal-plate-connected wood trusses are widely used in residential construction and continue to be increasingly used commercial and agricultural construction. A major reason for the success of this connection is the high mechanical performance and the efficiency in the production of prefabricated trusses using nail plates [1,2]. The increased use of nail plates with longer spans resulting in bulky trusses suggests that a more thorough understanding of the joint behaviour would be beneficial [3].

The main reference to the connection design is the European Standard EN 1995-1-1:2004 (EC5) which brings the Ultimate Limit State (ULS) equations to verify the plate design, but do not comprehensive information about the mechanical modelling to obtain these forces. The two ULS equations (8.52 and 8.55) from EC5 are based in the force distribution within the plate. The force is initially transferred from the timber to the nails by an anchorage area (Figure 1a), then from the nails to the steel plate. In the joint line (gap line) (Figure 1b), the net-cross-section transfer the force to the other side of the plate and finally to the other side of the joint (Figure 1c). Thus, two failure modes are identified: the capacity of the anchorage area to resist the distributed forces and the maximum load supported by the gap line section. Therefore, the design steps of a nail plate connection consist in determining the forces applied in the anchorage area and at the gap lines.



(a) First member anc. area (b) Single gap line (c) Second member anc. area

Fig. 1 - Nail plate force distribution

The present paper aims to contribute to the current research stage and present an numerical approach to evaluate the force distribution in joint sections with multiple gaps, since single gap sections are isostatic and can be solved manually.

LITERATURAE REVIEW

Some research efforts have been done on nail plate modelling using analytical [1,4] and numerical [5,6] methods. The main difficulty is to define the boundary conditions of the plate-wood contact region and the current approach is to use springs to simulate the teeth-wood interface, and the consideration of a non-linear spring combined with a rigid behavior in the gap line interface have been proven accurate to describe the load-slip curve results. The modelling should also account for the various rupture modes [7] – in-plane and out-of-plane – since the connection parameters can alter not only the ultimate stress value, but also the rupture mode.

One of the first FEM method was present by Foschi [4] and many subsequent researches are based on the following considerations. The 2D model assumed that the timber-plate interface was rigid, connected by non-linear springs. The stiffness of each spring, that simulated each "tooth" in the plate is evaluated according to mechanical and geometric plate properties. Load-slip was experimentally determined as:

$$P = \left(m_0 + m_1 |\Delta|\right)\left(1 - e^{-k|\Delta| m_0}\right) \tag{1}$$

Where P is load on the joint, Δ is joint slip, k is initial slope of the curve, m_1 is asymptotic slope of the curve and m_0 is intersection of the asymptotic slope with the load axis. Foschi's proposition successfully predicts max-load and slip curves, but the analysis was scoped to 2D failure modes, thereby tooth withdrawal and other out-of-plane modes could not be predicted.

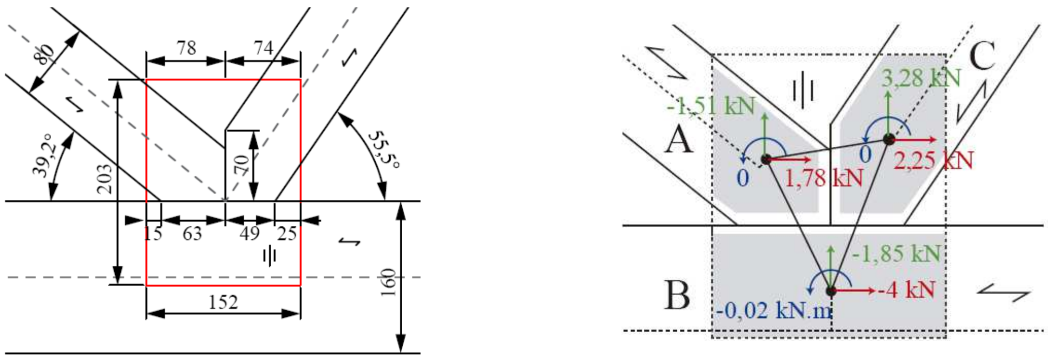
Cramer [1] also addressed the nail plate modelling, and took into consideration the joint eccentricity and the semi-rigid nonlinear behaviour of the connection. Each tooth was modelled as a group of three springs, two translational and one rotational, positioned at the centre of gravity of the tooth-plate contact area. The following rupture-deformation modes were studied simultaneously: tooth deformation, wood deformation and steel deformation Analysis has shown that the premise of the rigid behaviour of the gap line is reasonable to predict the force distribution within the gap line.

Groom [8] produced a theoretical model that accurately predicted the maximum limit states of different connection types. The method was based on the theory of elasticity and included the inelastic behaviour of the teeth and took into account the variation of the moment of inertia along the tooth length. The test approach was an incremental loading, where each load increment was processed assuming that at these intervals the joint behaviour was linear. The linear responses were then accumulated for each increment. The model was able to accurately predict the load-slip curves considering various types of geometric combinations.

Vatovec [9] used ANSYS software to analyse the connections. In this study, a nonlinear 3-D modelling was idealized, and each tooth was considered as a node composed of three nonlinear springs. The nail plate was modelled without gaps between the teeth. The model calculated the axial stress-strain curves accurately, but the authors reported that the rotational parameters were not validated due to lack of sufficient boundary conditions.

MULTIPLE GAP JOINT MODELLING

As previously stated, there are two EC5 equations that must be satisfied in terms of ultimate state that comes with the load distribution in two stages: the forces applied at the anchorage area and the forces in the gap lines. The first model that transfers the loads from the timber to the geometric centre in the anchorage area is documented on the third part of the French Standard DTU 31.3. The results given by the structural model are the loads acting on the anchorage area (Figure 2) and allow a straight forward anchorage area verification, therefore it will not be detailed here. This paper intends to provide further information in the gap line verification, by the analysis of the joint illustrated in Figure 3:



(a) Joint configuration

(b) Forces in the anchorage area

Fig. 2 - Nail plate study case

The objective is to transfer the loads applied at the geometric centre of each region to the gap lines, as shown in Figure 3. The system is pointedly undetermined, as each line requires three distinct force values (H, V and a moment M) that need to be solved simultaneously. Besides, there is no initial boundary conditions.

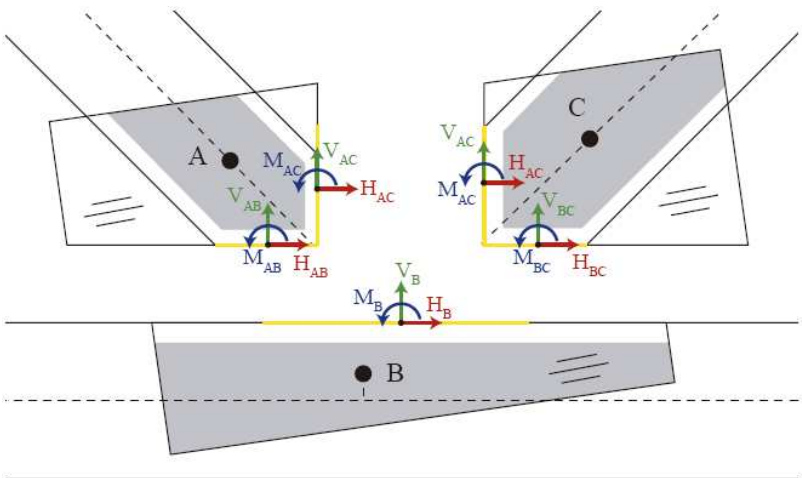


Fig. 3 - Force system in the gap lines

The proposed method is based in the premises stated by Foschi and Cramer. Initially, the line is divided by a finite integer number and to each new node, a set of springs are addressed. The springs will evaluate the slip in the nodes, and by multiplying by its stiffness, it is possible to extract a discrete value, that represents the distributed force in each node. There are three springs per node, to account for horizontal, vertical and rotational deflections.

The line behaviour is considered rigid, therefore there will be no relative rotation between two adjacent nodes. The rigid element consideration requires a formulation of geometric constraints or, as in most structural analysis software, a numerical approximation where the defined rigid elements are arbitrarily assigned with a proportional higher stiffness value. Another approach is to use rigid links, initially designed to simulate rigid diaphragm slab behaviour, which uses a geometric constraint between a group of nodes (slave nodes) in relation to a main node (master node). This method not only accounts for a precise rigid behaviour, but reduces the degree of freedom of the overall model, since the rigid members behave as boundary elements, not adding extra members and nodes to the system.

The spring stiffness is given by the slip modulus parameter K_{ser} which is a mechanical property of the nail plate. EC5 Table 7.1 gives reference values for the slip modulus of each connection type and provides a safety factor for the ULS slip modulus K_u :

$$K_u = \frac{2 \cdot K_{ser}}{3 \cdot \gamma_M} \tag{2}$$

Where γ_M is the partial factor for material properties. The translational and rotational stiffness are then defined:

$$K_x = K_y = 2 \cdot K_u \cdot A_{anc} \tag{3}$$

$$K_\phi = 2 \cdot K_u \cdot I_p \tag{4}$$

Where A_{anc} is the anchorage area and I_p is the anchorage area's polar inertia. With the given geometry in Figure 2, Table 1 presents the stiffness parameters for each anchorage area:

Table 1 - Spring stiffness of anchorage areas

	K_x [kN/cm]	K_y [kN/cm]	K_ϕ [kNm]
Area A	175,8	175,8	16,3
Area B	425,6	425,6	103,5
Area C	216,9	216,9	175,8

For a single area, the structural model is illustrated in Figure 4. The forces are applied in the centre of gravity of the anchorage area, which is the master node from the rigid link boundary element. For each node in the rigid link, a group of springs are associated, to account for the force distribution in the gap line.

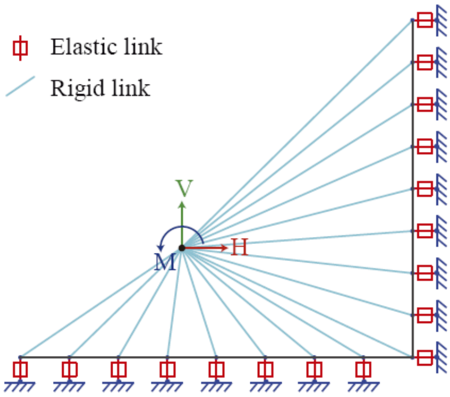


Fig. 4 - Single area structural model

This model can be used to every area in the connection, and given the Forces in the anchorage area in Figure 2b and the spring stiffnesses in Table 1, the following results were calculated (Table 2):

Table 3 - Force distribution by isolated areas

	V [kN]	H [kN]	M [N.m]
Line AB	0,750	-0,869	1,15
Line BC	-1,117	-1,662	1,02
Line AC	0,808	-0,933	-1,32

This is the common approach to solve the multiple gap system, using the rigid behaviour to the gap line and springs to account for the force distribution. Although this method could be applied to every area, the fact that adjacent lines may have different values depending on the area which the model is calculated, a generalized model is proposed. By taking the average stiffness for every adjacent line, it is possible to account for the whole connection system, as shown in Figure 5:

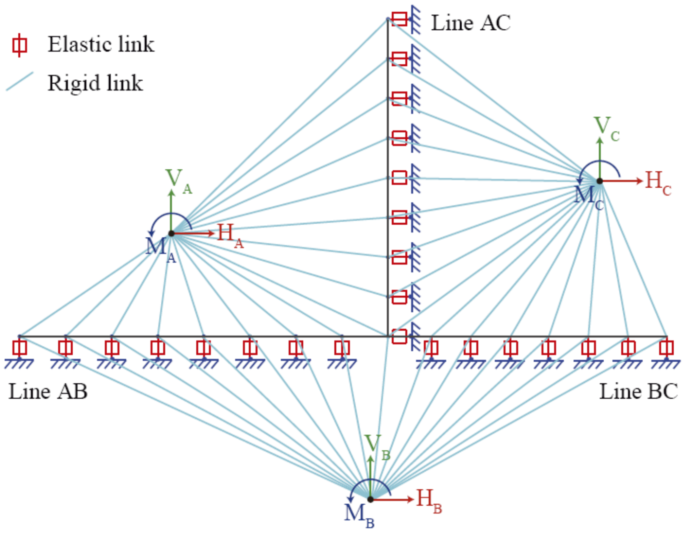


Fig. 5 - Multiple area structural model

The main difference in this method is the exploit of the overall force balance within the system. Since the internal forces in the plate must be in equilibrium, accounting for all the areas simultaneously will reduce the forces on each line, since adjacent regions with opposite forces will now be part of the same global system.

The spring stiffness shall be adjusted, taking account the anchorage area influence in adjacent lines. For each line, the effective stiffness will be the average between the adjacent areas (Table 3):

Table 4 - Effective spring stiffness

	K_x [kN/cm]	K_y [kN/cm]	K_ϕ [kNm]
Line AB	300,7	300,7	59,9
Line BC	321,3	321,3	139,65
Line AC	196,4	196,4	96,05

The force distribution values are presented in Table 4:

Table 4 - Force distribution by multiple areas

	V [kN]	H [kN]	M [N.m]
Line AB	-1,915	0,563	0,17
Line BC	0,599	0,904	1,02
Line AC	0,11	-0,623	-0,33

As expected, the overall force values were lower compared to the isolated areas due to the equilibrium of anchorage areas. The biggest reduction was noted in the bending moment value, 46%, which is a satisfactory result, since bending moments are often the design limitation in truss rafters.

CONCLUSIONS

In order to solve the multiple gap problem, it was necessary to impose boundary conditions on the gap lines. The main theory is the consideration of a rigid behaviour for the gap line and a set of springs to account for the force distribution. The first approach evaluating each area individually led to overestimated values, since the independent forces on each area don't contribute to the system's equilibrium. The second approach using multiple areas provided a simplified method to evaluate the whole connection by averaging the spring stiffness of each adjacent lines. Since the current standards make no mention on how to obtain the gap line forces, it is intended to use experimental tests to select and correct the numerical model.

REFERENCES

[1] Cramer SM, Shrestha D, Fohrell WB, "Theoretical consideration of metalplate- connected wood-splice joints". In: Journal of Structural Engineering 116.12 (1990), pp.3458-3474.

[2] Callahan Edward E, Dinsmore Pamela W, Metal Plate Connected Wood Truss Handbook: A Comprehensive Guide to the Design and Use of Metal Plate Connected Wood Trusses in Construction Today. Wood Truss Council of America, 1993.

[3] McMartin KC, Quaile AT, Keenan FJ, "Strength and structural safety of long-span light wood roof trusses." Canadian Journal of Civil Engineering 11.4 (1984): pp.978-992.

[4] Foschi Ricardo O, Analysis of wood diaphragms and trusses. Part II: Truss-plate connections. In: Canadian Journal of Civil Engineering 4.3 (1977), pp.353-362.

[5] Misra RD, Esmay ML, Stress distribution in the punched metal plate of a timber joint. In: Transactions of the ASAE 9.6 (1966), pp.839-842.

[6] Gebremedhin KG, Jorgensen MC, Woelfel CB, Load-slip characteristics of metal plate connected wood joints tested in tension and shear. In: Wood and fibre science: journal of the Society of Wood Science and Technology (1992).

[7] Lau Peter WC, Factors affecting the behaviour and modelling of toothed metal plate joints. In: Canadian Journal of Civil Engineering 14.2 (1987), pp.183-190.

[8] Groom Leslie, Polensek Anton, Nonlinear modelling of truss-plate joints. In: Journal of Structural Engineering 118.9 (1992), pp.2514-2531.

[9] Vatovec M, Miller TH, Gupta Rakesh, Modelling of metal-plate-connected wood truss joints. In: Transactions of the ASAE 39.3 (1996), pp.1101-1111.