Development of an optimized moment-connection joint for pultruded profiles

Jessé Henrique Nascimento Beserra[1], Daniel Carlos Taissum Cardoso[1], Jorge Lopes Santos[2]

[1]*Dept. of Civil and Environmental Engineering, Pontifical Catholic University of Rio de Janeiro (PUC-Rio), Rua Marques de Sao Vicente, 225, Rio de Janeiro, RJ,22451-900, Brazil*
jessehnb@gmail.com, dctcardoso@puc-rio.br
[2]*Dept. of Arts & Design, Pontifical Catholic University of Rio de Janeiro (PUC-Rio), Rua Marques de Sao Vicente, 225, Rio de Janeiro, RJ,22451-900, Brazil*
jorgelopes.int@gmail.com

Abstract. The present work aims the development of a novel optimized beam-column moment connection for pultruded profiles that is characterized by an efficient shape able to transfer load between members due to contact and friction. To accomplish this task, the work will be divided in two major stages: i) stress analysis via finite element model; and ii) topology optimization. In the former step, a reference joint geometry will be pre-designed and stress trajectories are obtained for a given load using software Abaqus. Then, the joint shape is optimized using a topology optimization technique available in the software package. During the study, stresses are evaluated and the behavior of both optimized and original joints are compared in the elastic range. This work is part of an ongoing investigation on the development of novel 3D-printed solutions for joints.

Keywords: pultruded, composite, connection, optimized

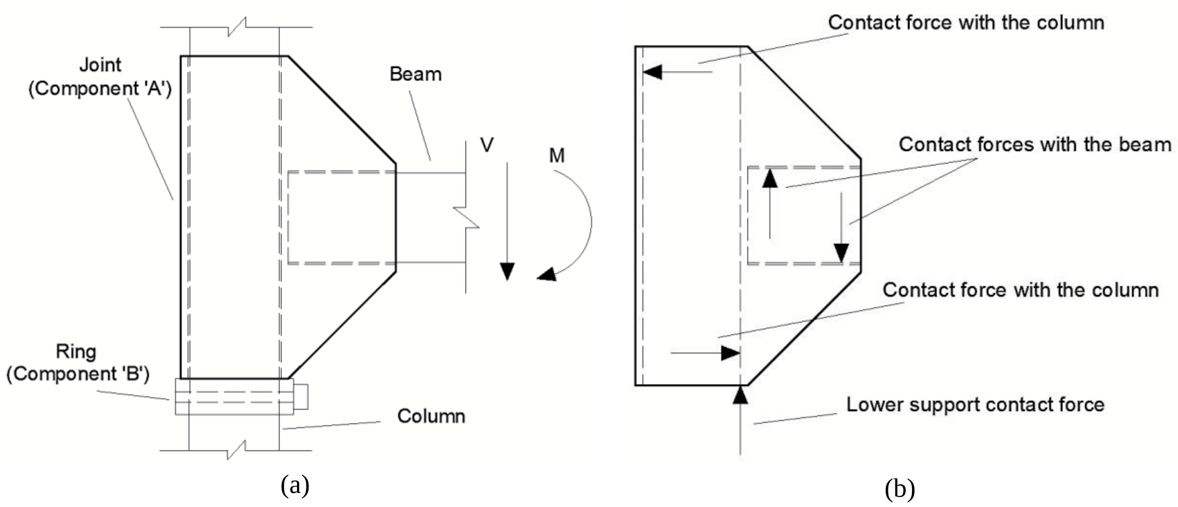
1 Introduction

Pultruded profiles, especially those made by glass-fiber, are characterized by low elastic modulus, high orthotropy ratio and brittleness. However, they have high strength-to-weight ratio and are the most common option regarding FRP applications within construction industry due to its competitive cost. In structural systems, bolted joints are usually the preferred option for transferring load between members, although discontinuities due to the holes in combination with the aforementioned material properties become a critical matter for load carrying capacity. Another challenge concerning pultruded profiles joints, but less discussed in literature, is the design of rigid and semi-rigid joints, since the connection components have low stiffness. To address these problems, a new approach for pultruded joints is attempted, comprising the development of an optimized component capable of transferring moment without requiring any drilling on GFRP.

2 Initial design

2.1 Conception

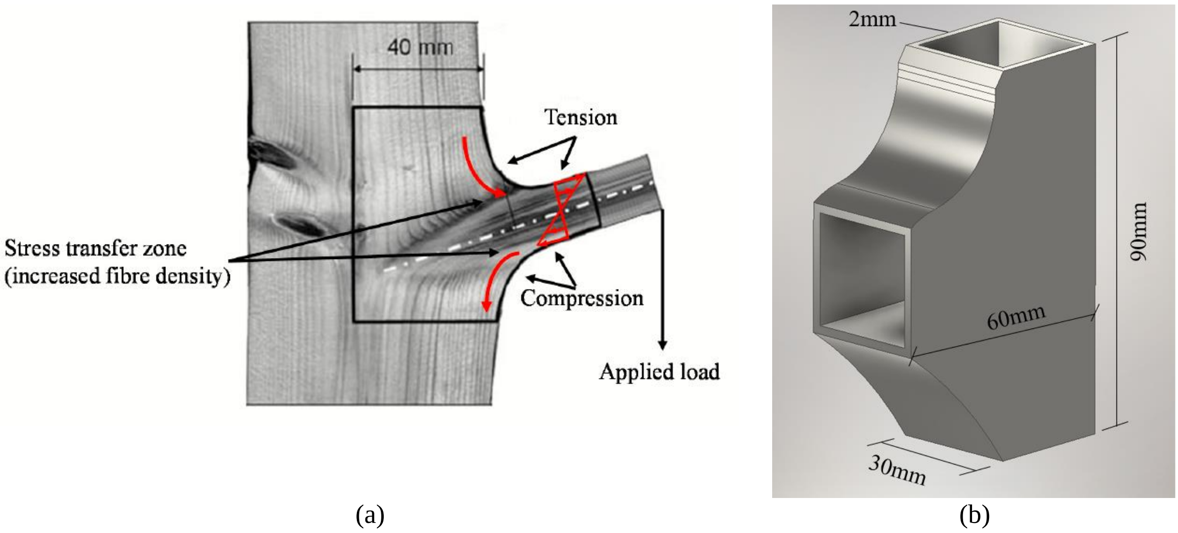
In order to establish a basic geometry for the connection, it was considered a simple two-dimensional beam-column load case. For this specific study, two squared tubular profiles have been chosen as the structural members to be connected and, as a first geometry approach, a symmetric component ‘A’ with triangular transition regions has been defined, as shown in Fig. 1a. This component is responsible for transferring the moment. To provide vertical support, a ‘ring’ component ‘B’ is positioned below ‘A’; this component must be adhesively bonded or adequately pressed against the column with tightened bolts for a friction connection. The study of component ‘B’ is out of the scope of the present study.



(a) (b)

Figure 1. Initial joint design: (a) First geometry approach and (b) Contact forces acting on the component ‘A’.

Such a geometry was inspired in typical moment resistant steel joints and some GFRP beam-columns connections designed by Bank [1] and Mottram [2]. It is worth noting that the component does not need to be symmetric with respect to the beam axis, once the high-loaded paths depend on the loading configuration shown in Fig.1b. Furthermore, another way to analyze this symmetry problem is by applying bionics principles and taking it similar to a trunk-branch joint in a tree, where the trunk mimics the column and the branch mimics the beam, see Fig. 2a, obtained from Avgoulas [3]. Looking closely at this type of natural joint it is possible to see that the lower part has a smoother curvature, whereas the upper part has a smaller radius. Thus, considering these two remarks, a basic geometry could be achieved for the moment-connection joint, as shown in Fig.2b. It is important to mention that the dimensions shown here are small because it corresponds to those that are going to be applied for a 3D-printed prototype.

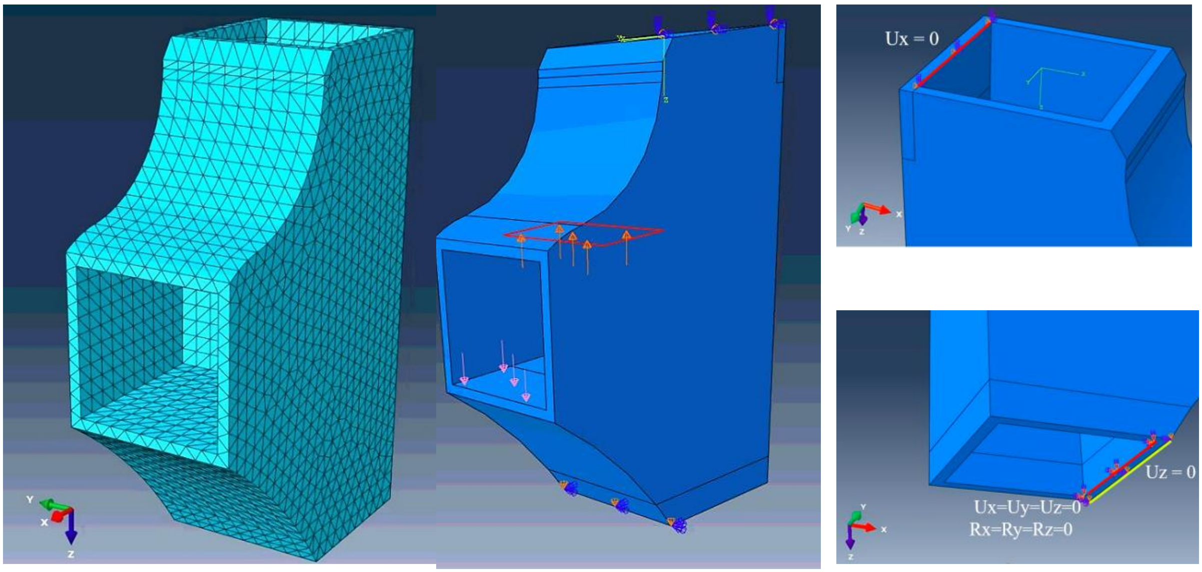


(a) (b)

Figure 2. Forces and stress distribution acting on a branch-trunk joint (a) obtained from Avgoulas [3] (b)Basic geometry defined for the moment-connection

2.2 Stress Analysis

Once the basic geometry and the load case have been defined for the joint, an elastic stress analysis can be carried to verify the behavior of the component in terms of stresses and strains. To do that, a finite element model has been created using Abaqus software. The material used on the simulation was an isotropic Acrylonitrile Butadiene Styrene (ABS) with 2.5GPa elastic modulus, 0.45 of Poisson ratio and 1030kg/mm³ of density, according to the 3D-printed prototype to be fabricated. To mesh the part, a simple stress convergence study has been conducted and, further, a free meshing with a quadratic tetrahedral element (C3D10) at an approximate global size of 3mm was selected, combining good accuracy and short processing time. This resulted in a total of 14638 elements and 26322 nodes, see Fig. 3a.



(a) (b) (c)

Figure 3. Moment-connection joint: (a) Mesh and (b) Loading condition (c) Boundary Conditions

The load condition considered in the analysis consists of a moment resulting from a binary, which is obtained by applying a uniform pressure of 2MPa on two internal surfaces of the connection, as shown in Fig.3b. For the boundary conditions, simple support links have been defined, see Fig. 3c. The lower part of the joint had two restrained edges, one for displacements along z axis (shown in yellow in Fig.3c) and other (highlighted in red in Fig 3c) subjected to an encastre, just to maintain tridimensional equilibrium. The upper part of the joint, especially the edge highlighted in red in Fig. 3c, was constrained for displacements along x direction.

3 Topology Optimization

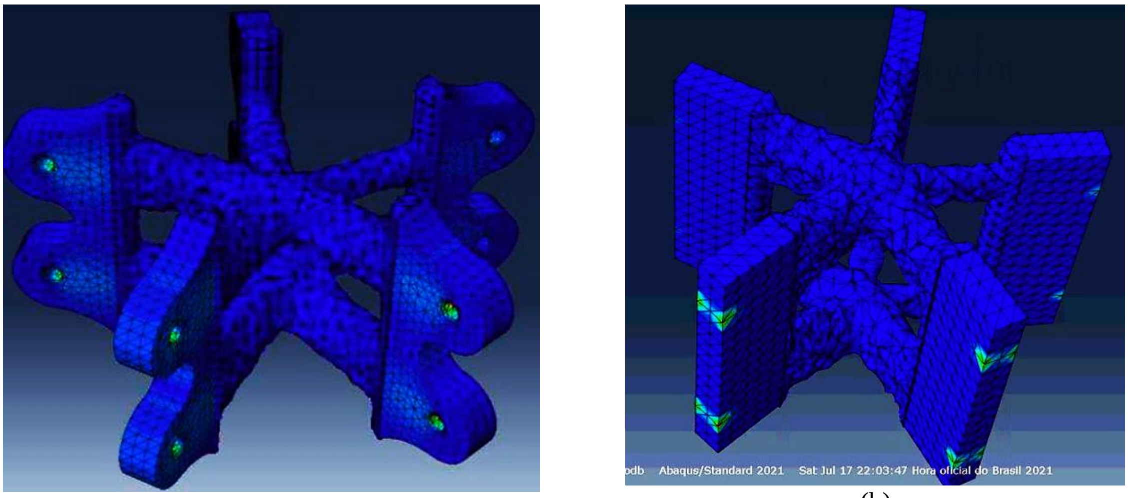
3.1 Validation

Before performing the topology optimization problem of the current work, it is important to validate the optimization technique. To accomplish that, a study carried by Abdelwahab [4] was used as a reference/benchmark. In the aforementioned work, the authors optimized a cylindrical-shaped truss joint (see Fig.4) for four different loading cases. For the sake of simplicity, only one load case (shown in Fig.4b) has been used herein.

(a) (b)

Figure 4 Cylindrical-shaped truss joint studied by Abdelwahab [4] (a) basic geometry; (b) load case

Similarly to Abdelwahab [4], quadratic tetrahedral elements (C3D10) were used to mesh the part and the elastic properties applied for steel were 210GPa for elastic modulus, 0.3 for Poisson ratio and 7800kg/m³ for density. However, the mesh refinement applied on the referred work was not replicated here, once it comprises 148438 elements and 212127 nodes, thus, demanding a high computational cost. Instead, a mesh composed by 40533 elements and 59710 nodes was applied to the model. Regarding the objective function of the optimization task, the main goal of the authors was to minimize the strain energy of the joint whilst keeping a volume fraction constrained to 53.1% of the initial value of the joint. Although, the geometry used here as an input for the optimization did not have the same volume of the real joint, so, a reference of 21.5% was estimated after some calculations and the convergence between the results was obtained by trial and error. The loading magnitude applied to the model was half of that used by the authors, because the full loading promoted singularities and led to a non-converging optimization. In order to secure the equilibrium of the model, one of the applied forces was replaced by an encastre support, which was the only boundary condition applied to the model.



(a) (b)

Figure 5. Results obtained by (a) Abdelwahab [4] and (b) the current work.

In Fig 5a and 5b the topology optimization results obtained by Abdelwahab [4] and the current work are exposed side by side, respectively, and it is possible to infer that a good convergence has been achieved. In this case, the volume fraction constraint was set to 28%, which was the smaller value that still provides a converging model.

3.2 Optimization Procedure (or Process)

To carry the topology optimization, the mesh, loads and boundary conditions were kept exactly the same as the elastic stress analysis, so, all the information of this section also applies to the current one. So, the first task is to define the design region, that is, select the domain of the optimization algorithm. For this specific study, the whole model has been selected as design region, once it is intended to define the volume fraction in terms of total volume of the joint. After that, a condition-based algorithm is selected for the optimization, because, for simple problems, it is more efficient than the general algorithm. Concerning the objective function, the main target was to maximize stiffness (minimize strain energy) while constraining the volume to a fraction of the initial value. In this context, it is worth mentioning that two volume fractions were selected, namely, 70% and 50% of the initial volume. In order to keep the model symmetric to the xz plane, a geometric constraint for symmetry along y axis was applied to the optimization task, furthermore, some areas have been maintained, just to preserve contact regions, as shown in Fig. 6.

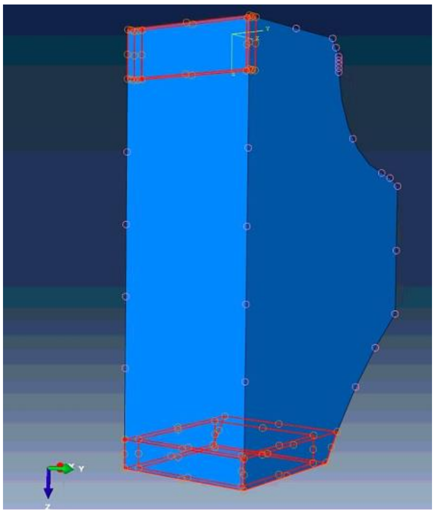
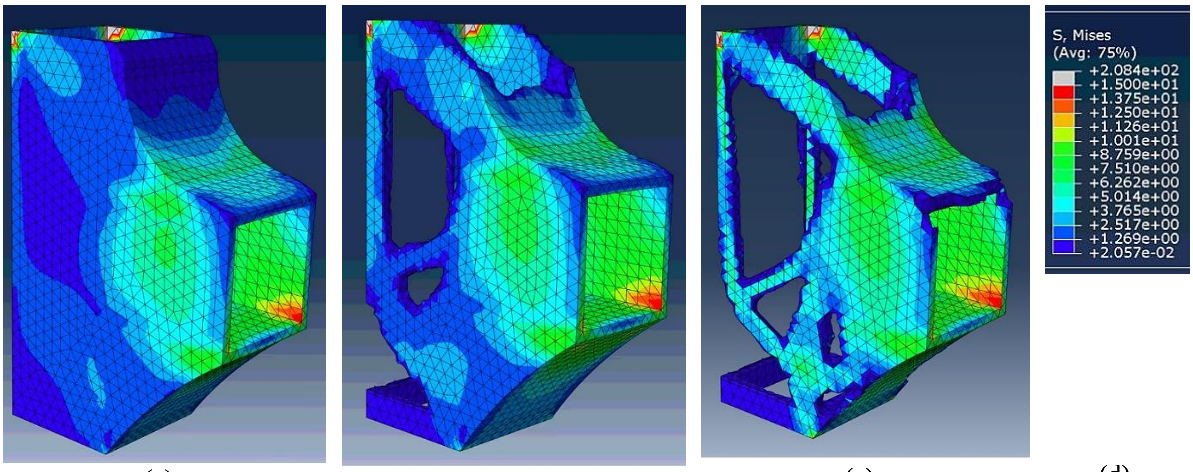


Figure 6. Frozen regions of the optimization procedure

4 Results

With the stress analysis and the topology optimization successfully completed, see Fig 7a to 7c, it was possible to note that in all cases the highly stressed region (shown in red) occurs at the same location, namely, the neighborhood of the downward force coming from the beam. Further, it is worth mentioning that, for both two optimized geometries, shown in Fig.7b and Fig.7c, the lower part of the component seems to be critical for ultimate failure load, since it comprises stability matters. This statement is based on the fact that this region, which is highly subjected to compression, had a high amount of material removed during topology optimization, becoming almost “hollow”.



(a) (b) (c) (d)

Figure 7. Stress distribution obtained for (a) original joint and optimized joint with volume fractions of (b) 70% and (c) 50% (d) Stress spectrum

Figure 7d exposes the spectrum of Von Mises stresses, ranging from 0 to 15MPa, obtained from the elastic analysis on Abaqus for both three geometries. Thus, it is possible to see in Fig. 7a that the original geometry has a significant part with stresses close to zero, which can explain that, even with a high reduction in volume, the highest stresses were nearly the same between optimized and original joints, as can be seen in Fig 7a and Fig 7c. It is important to point out that the optimized geometries also had a small part with almost no stresses, however, in these cases, the region resulted from the need to keep the correct position between joint and column, enabling a proper contact and assembling.

5 Conclusions

In this work, an optimized moment connection joint for squared tubular pultruded profiles has been designed. To accomplish that, a topology optimization process was applied to an initial basic geometry, enabling a 50% reduction in volume without compromising the performance of the component in terms of stresses and strains. This geometry will be used as a reference for the development of 3D-printed prototype. It is worth noting that stability problems may arise as the volume reduction increases, since the compressed region becomes thinner. The results herein presented will be further validated with experimental data and more refined FEM models. Also, for the real-size joint, it is intended to apply glass fiber reinforcement by lamination, so, as fibers are more efficient when subjected to axial tension loads, the tension and compression load paths must be carefully investigated in the future.

Acknowledgements. The authors would like to thank PUC-Rio, LEM-DEC, FAPERJ and CNPq for the technical and financial support.

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owner Konstantinos Tsavdaridis [4] and Michael Sutcliffe [3].

References

[1] L. Bank, *Design and performance of connections for pultruded frame structures*. Journal of Reinforced Plastics and Composites, 1994;13
[2] J. Mottram, *A review of the behaviour and analysis of bolted connections and joints in pultruded fibre reinforced polymers.* Materials and Design, 2015
[3] E. Avgoulas, *A Review of Natural Joint Systems and Numerical Investigation of Bio-Inspired GFRP-to-Steel Joints*, 2016
[4] Abdelwahab & Tsavdaridis *Optimised 3d-printed metallic node-connections for reticulated structures*, 9th International Conference on Steel and Aluminium Structures, 2019

.