Proceedings of the IASS Annual Symposium 2019 - Structural Membranes 2019
Form and Force
7-10 October 2019, Barcelona, Spain
C. Lázaro, K.-U. Bletzinger, E. Oñate (eds.)

Segmented Spiral Using Inter-Connected Timber Elements

Petras VESTARTAS [*,a], **Loic PELLETIER** [b c], **Martin NAKAD** [a], **Aryan REZAEI RAD** [a], **Yves WEINAND** [a]

[*,a] Ecole Polytechnique Fédérale de Lausanne (EPFL), Laboratory for Timber Constructions (IBOIS)
EPFL ENAC IIC IBOIS, GC H2 711 (Bâtiment GC), Station 18, CH-1015 Lausanne, Switzerland
petras.vestartas@epfl.ch

[b] Université du Québec à Chicoutimi (UQAC), QC G7H 2B1, Canada
[c] Ecole Nationale d'Ingénieurs de Saint-Étienne (ENISE), 42100 Saint-Étienne, France

Abstract

This paper describes the construction process of a scenographic structure - five meters high spiral. It consists of following parts: CAD modeling work-flow, prototyping and FEM structural calculation, assembly and on-site loading test. The structure was composed of twenty-nine wooden boxes, which were screwed together and suspended by cables. The custom FEM model was made to understand the structural behavior of the spiral. The principal objective of structural calculation was to understand the rigidity of the connections and the interactions between the wooden-boxes, in order to apprehend the behavior of the structure under loads. Further details are given about different steps involved in modelling and fabrication, such as parametric tools, G-Code tool-path generation for the 4.5-axis CNC machine and on-site installation. Afterwards, the loading tests and point-cloud deflection comparison are discussed in order to highlight the possible optimization of such structures.

Keywords: Cross-laminated timber, wooden spatial structures, FEM, CNC, Comparing Point Clouds, digital fabrication, design to assembly, wood connections, integral mechanical attachment

1 Introduction

The spiral was made of CLT 3-plys spruce panels for the boxes, and steel cables to resist the tension loads. The interior of the cathedral was scanned using Faro Model S 150 in order to have accurate measurements for the project. The figure 1 shows the scanned church model and project including the stands with a constant slope in the direction of the stage, and the spiral positioned at the center of the cathedrals nave. Above the spiral, the cables are fastened to the framework, which stands thirty meters higher.

Automated CAD Design The use of digital fabrication and automation of the fabrication process was required, because of the precision and the uniqueness of every parts of the structure. The 3D modelling algorithm was developed for creating box-to-plate assembly (figure 2), depending of the initial inputs, such as cone height and width, the thickness of the panels,

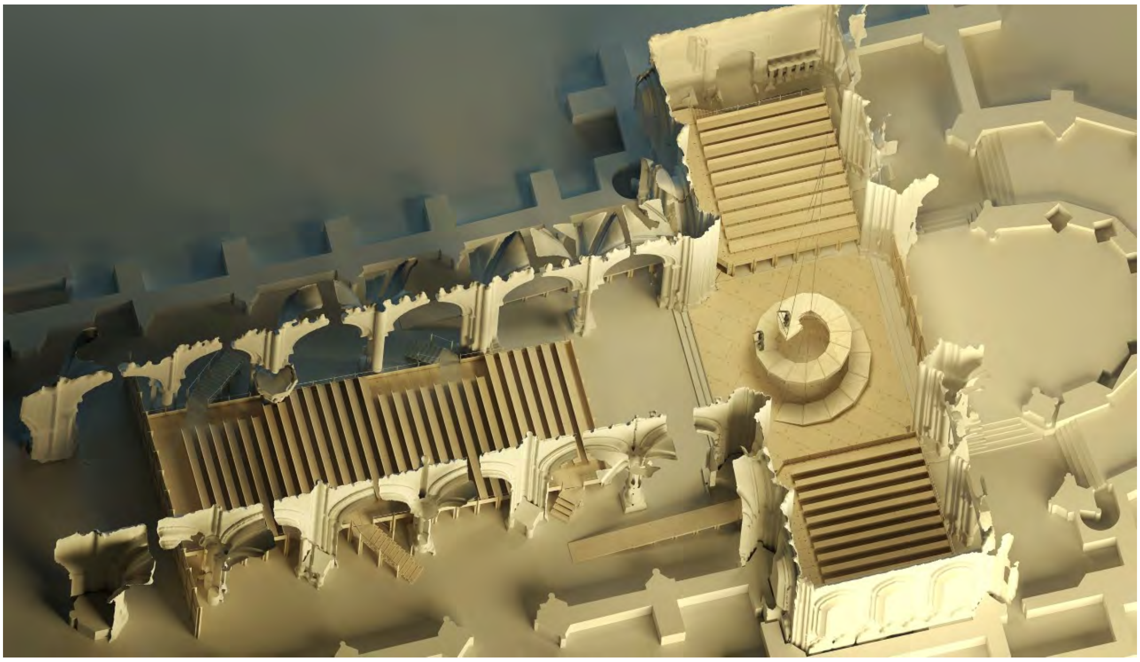


Figure 1: The three stands with a constant slope in the direction of the spiral.

joinery (finger-joints), external connectors (dowels, screws.), assembly sequence and fabrication tool-paths.

Finite Element Modeling The structural model was built in a Finite Element computational platform ABAQUS for analysis purposes. The FE model was then used for the design procedure, where the global stress values and the displacements were calculated. Following the different load cases, the structure demonstrated acceptable level of performance. Nevertheless, its global performance could be improved by changing its initial geometry.

Computer Numerical Control Machine 4.5-axis CNC machine "Maka" was used for cutting panel outlines, drilling holes for screws and dowels, engraving panel indices and cable inlets. Furthermore, the G-Code tool-path was automated from design to fabrication within the same software interface. The whole cutting process took approximately two weeks.

3D Laser Scanning and Point-Cloud Deflection Comparison At the end of the project, the entire structure was scanned to measure the deformation, with different loading cases scenario. First, we aligned two point-clouds by picking equivalent point of each scan and then computed the distance between each point to know the deformation.

2 Design and CAD: Geometry

Conic Shape The scale of the spiral was chosen by minimal bounding area of central vault. The geometry of the spiral is derived from a cone shape, whose central axis and angle is used to position stairs within user specified angle/stair width (figure 2A). Moreover, this approach guarantees to keep an unique point of intersection for the cables, at the top of the structure.

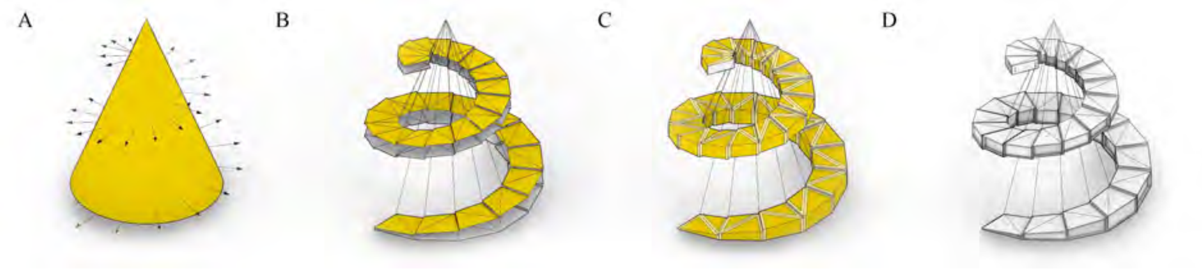


Figure 2: (A) The spiral steps follows the vectors along the cone (B) Base model: two layers of steps moved by z-axis (C) Model for structural calculation (D) Final model with thickness, joinery and tool-path

Steps Geometry The model is divided into two top and bottom layers that determine the static height of the structure. Also, the model is divided vertically into three parts: bottom ramp, middle sloped steps, and top equal height deck. The number of steps was determined by the minimal slope of overall gradient of spiral ramp. In addition to this, the step is sloped to reduce its height, and triangulated to keep them planar. The first three panels are weighted and screwed into a floor made out of 25 mm thick Oriented Strand Board (OSB) panels. Each corner of this floor is contiguous to the piles of the cathedral, and avoid the lateral displacement in this plane for the entire structure. The main issue could have been the instability of the structure caused by a dynamic load on it, like singers. The first three boxes could have lift up from their position and slipped, because the main cable could also deviate.

Detailing and Fabrication The low-poly mesh model is used for fast design iterations and structural calculations (figure 2 B-C). Afterwards the model is processed introducing fabrication constrains: thickness of panels, assembly sequence, joinery (finger-joints and screws), labeling, orientation to CNC machining space and G-Code tool-path generation (figure 2D). The geometry of spiral was divided into box objects, that contained information about next and previous boxes. The adjacency of boxes gives information about thought-tenon joint, screws holes and dowel positioning joints derived from neighbours. Each box class contains 9 plate elements: two connecting plates, two sides that have different inlet to have space for screwing, 1 diagonal plate flipped to triangulation of step for better structural performance and 4 triangular elements (figure 3). The collection of plates contains adjacency of plate-to-plate assembly within one box

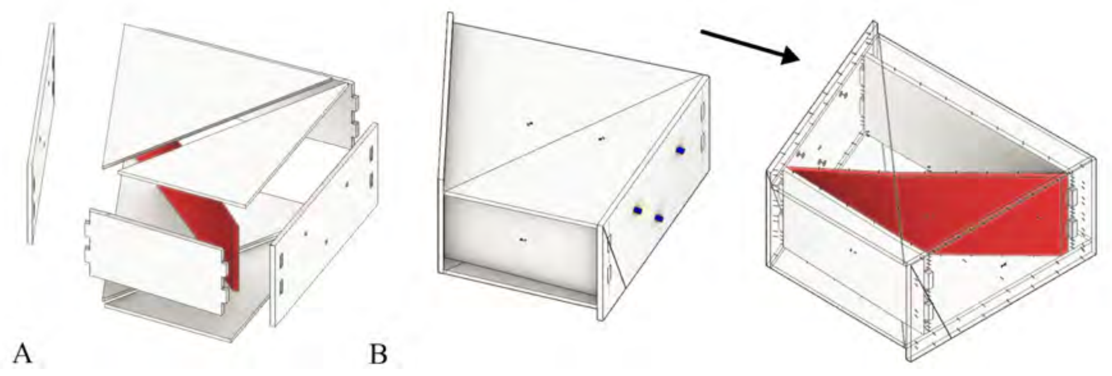


Figure 3: (A) Exploded view of a box (B) Box with dowels and engraved (C) Box with screws

to indicating screwing holes in the CNC machine and tenon-mortise joints.

3 Prototype

3.1 Structural test

An initial test was conducted in the laboratory to validate the boxes configuration, the position of the different elements, the simplicity of the assembly on site and the response under loads. Four boxes were fabricated and were loaded in cantilever to measure the deformation and examine the weaknesses (figure 4A). The first box was fastened to a wooden base made of a 25mm OSB panels with three rods, which was fixed to the ground using sixteen rods connected into the concrete slab. This configuration avoided the boxes from tilting under the loading, and enable to put more weight on them to observe further deformations. The first phenomenon was the rotation between the part of the box close to the interior of the curve, and the exterior one. The second phenomenon was the emergence of cracks between the first box and the second one (figure 4B). The structure provided many information which were crucial to anticipate potential future problems by observing and measuring the displacements on the last caisson in cantilever. Eleven LVDT sensors measured the displacements of the boxes horizontally and vertically during the loading. The loading was applied using 11.9 kilogramms weights every 30 seconds on the structure, and the loaded box had a vertical displacement of 55 millimeters at 319.45 kilograms. A total load of 408 kilograms was applied to make the structure collapse and observe behavior of proposed system. The weakest part is the junction between the fixed part and the cantilevered one. After the test, the delamination of side panels was visible. This data was used to make a Finite Element Method based on the Abaqus calculation software, which took into account the rigidity of our elements and the connectors between them.

3.2 Finite Element Method

A Finite Element Method model was developed to support both the experimental test and design framework [1][2][3]. The displacements and stress values of the elements are calculated and compared to the timber mechanical characteristics and associated limits (figure 4C). Specifically, the stress in the metallic elements, as well as the cables were compared to the ultimate and serviceability limit state. Considering the real load cases, a uniform distributed load of 1 kN was applied at four boxes. A maximum displacement of 10 millimeters at the top of the spiral was observed which meet the SLS limit. Furthermore, the FE model played an important part for the choice of the different metallic parts, where it made the adaptation of the quality and

Figure 4: (A) The 4 boxes prototype (B) The failure mode (C) FEM calibrated after the test

the size section possible.

3.3 Optimization of the caissons

Following the previous results and the assembling of the prototype, changes were done to upgrade the current model:

- Implementation of a diaphragm diagonally inside the caissons: allows a better rigidity of the caissons and between them (figure 3A).
- Dowels between the boxes : the boxes were lifted and positioned at the correct place against the previous one using 3 dowels at each side. They were not designed to withstand shear forces, but only to guarantee the precise alignment (figure 3B).
- Pre-drilling for screws: during the assembly of the prototype, the time necessary for screwing was important comparatively to the others tasks. Pre-drilling the holes during the CNC fabrication phase allowed a faster installation on site. It also prevented from mistakes and provided better connections (figure 3C).

4 Fabrication and Automation of G-Code

The automation of geometry generation included tool-path generation for CNC machine "Maka" (figure 5E). Firstly, the two-hundred and sixty three elements of the spiral are flattened at the same level (figure 5A-B) and placed on machining space of the design software. The nesting of oriented plated were performed using open-source OpenNest Grasshopper add-on (figure 5C). The custom G-Code tool selects geometrical elements such as lines drilling, outline for cutting, text for engraving and sends the G-Code for CNC fabrication (figure 5D). This step optimized the file preparation and avoided errors in programming once it is properly made. The code only needs to be checked visually on the screen with the 3D-simulation of the cutting, and then start the machining.

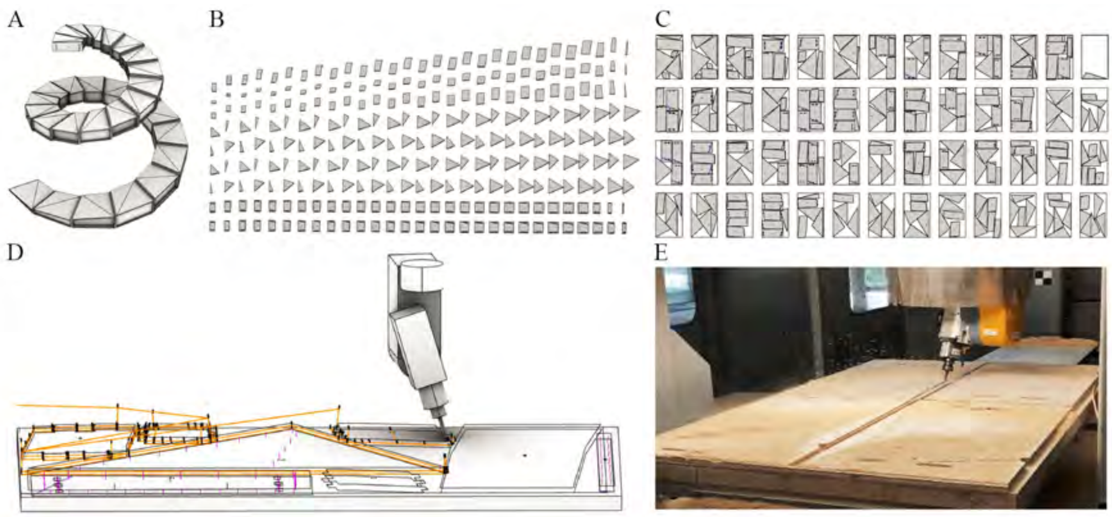


Figure 5: (A) Complete spiral (B) Flattened elements of the spiral (C) Nesting of oriented plate (D) 3D-simulation of the cutting machining (E) Maka CNC at work

The custom code was also developed for surfacing the angles superior to 45. The cutting tool was too small and the rotor of the CNC-machine could collide with the panel and de-calibrate the machine. To avoid this problem, the position of the diaphragms has been changed to have a smaller angle, and the angles were cut by surfacing the panels with a slope instead of cutting them through. The whole geometry and G-Code application were modified quickly thanks to automated design. Moreover, all the cutting angles of plates must be feasible from one side of the panel, because the CNC-machine can work only on one side of the panel and flipping the panel was not feasible.

5 On site installation method

The on-site assembly took three days. Firstly, the main cable was fastened to the frame of the cathedral, that also determined the center of the scene thanks to a plumb line. When the center was marked, the position of the first box was drawn on the floor. Afterwards, the first three boxes were self weighted and screwed to the OSB panels flooring. The boxes were lifted using hoist which was fastened at the end of the main cable and put against the previous one. The box components were correctly positioned by three dowels, screwed and shored directly, as it is shown in the figure 6A. Measurements were made to check the height of each box, assuring a correct re-partition of the tension into the cables. Once every box was shored, the cables were fastened by threaded rods and then the complete spiral was fixed as it is shown in the figure 6B. Measurements were made again to check the height of each box and if necessary cable length was modified by putting tension into the cable.

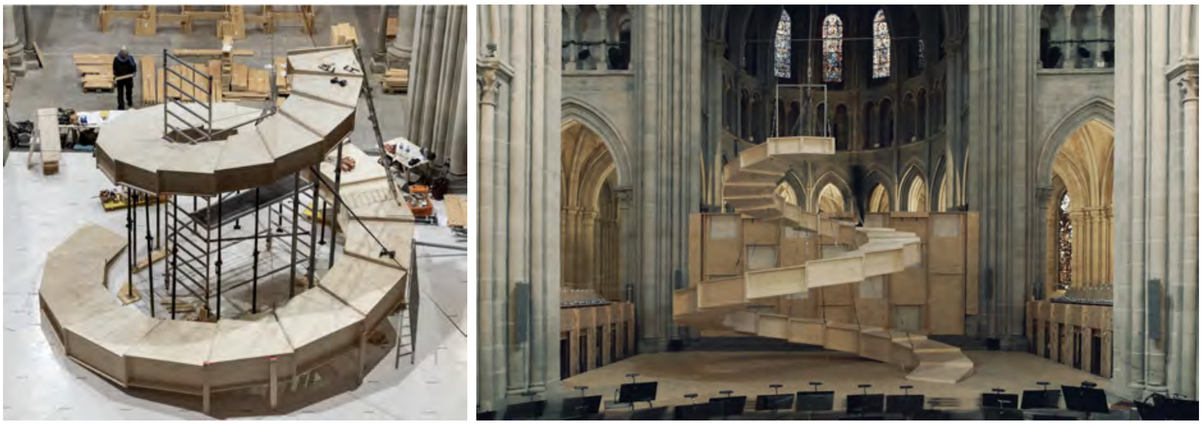


Figure 6: (A) The shored spiral during assembly (B) The hanging spiral [Credit: Claude Bornand]

6 Structural test of the complete spiral

6.1 Loading test and 3D laser scanning of the deflection

The structure was tested on-site after all the events took place (for safety reasons). Four different loading cases were applied on the spiral to study its deflection and global behavior under loads. The loads simulated human weight on different part of the structure. In order to better understand the position of the loading, the boxes were numbered as shown in the figure 7A. A laser scanner was used to measure the displacements during the different loading cases [4][5]. Six scans were made in total:

- Scan 1: Initial position : Get the real position of the spiral as a base for comparison
- Scan 2: Loading 1: 95.2 kg on the box 19.
- Scan 3: Loading 2: 95.2 kg on the box 19 and 13 (190.4kg in total).
- Scan 4: Loading 3: 95.2 kg on the box 19, 13 and 5 (285.6kg in total).
- Scan 5: Loading 4: 285.6 kg on the box 19.
- Scan 6: Final position : To see the remaining displacement after the load

The results of the different loading cases are shown on the figure 8B with the FEM comparison. For the first loading case, we observed positive displacements on the opposite side of the loaded area. This was explained by the fact the metallic plate, which withstand the cables, rotated with the loading, in addition the structure is unbalanced by the loading too. The second loading case applied a new load on the opposite side of the spiral but closer to the center, which result in a smaller deflection on the box 19 than for the loading case 1. For the third loading case, as shown on the figure 7A-B, we observed a bigger displacement on the most loaded side of the spiral, showing the transmission of the deflection by the cable, between the box 5 and 19. During the fourth loading, we hear some cracking noise in the junction between the start of the cantilever (Box 25) and the third box attached to the OSB flooring (Box 26). This observation validated the prototype test results, when the most loaded part of the structure is between the fixed part and the cantilevered.

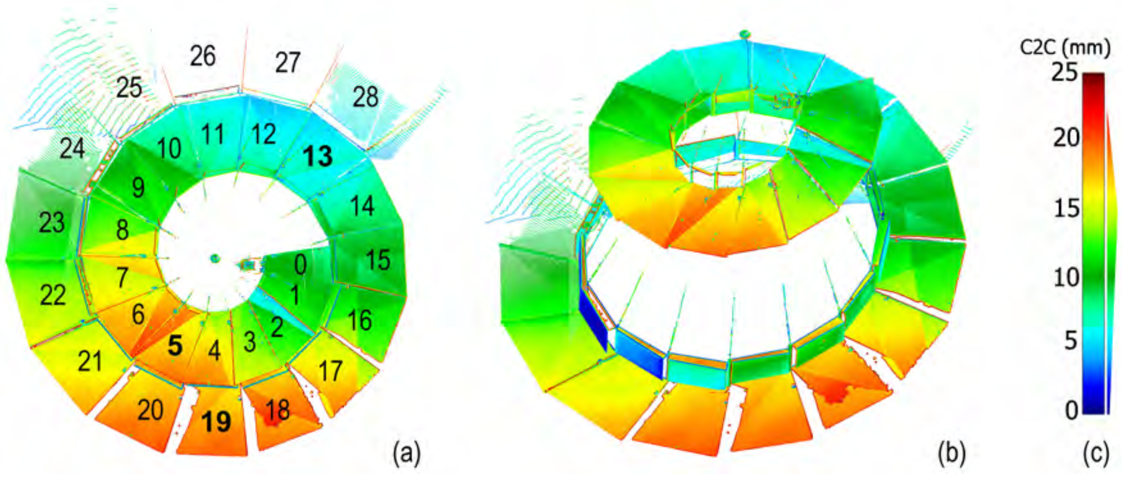


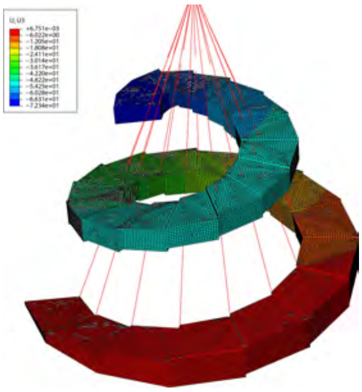
Figure 7: (A) Loading case 3: Top view (B) Loading case 3: Perspective view (C) C2C values

6.2 Comparison with Finite Element Method model)

The initial FEM model is then computed with the same loading as it was in the experimental test to compare the results. The model give a maximum of displacement of 9.25 millimeters against 10.9 millimeters in reality. The global behavior of the FEM-model followed the real case.

7 Conclusion

The project served as the main opera stage design applied for a specific event (figure 6B). It was also a first demonstrator when the laboratory experimented with timber plates structure in cantilever action. Several developments were made and would be applicable to other projects



Case	Box	Load	FEM	C2C
	No	[N]	[mm]	[mm]
1	19	934	9.25	10.9
2	19	934	3.12	4.3
2	13	934	15.4	17.9
3	19	934	16.65	19.9
3	13	934	5.72	7.6
3	05	934	16.75	18.7
4	19	2802	28.31	29.4

Figure 8: (A)FEA of the spiral under the loading case 3 (B) Comparison C2C and FEA results

including G-Code, nesting, and geometry generation by means of box-to-plate data-structures that contained joinery information. The project used automated work-flow from design to fabrication which was efficient in terms of short cutting and assembly time. Moreover, we employed collaboration from CAD to FEM through low-poly models at early stages of design. The same model served as an input for detailed fabrication model and structural analysis application. Nevertheless, the work-flow could have benefited more from integral mechanical attachment in order to reduce the amount of screws used in this demonstrator. Finally, the multidisciplinary team of designers, engineers and fabricators worked together in order to improve and realize this temporary free-form timber application.

8 Acknowledgements

The authors would like to thanks EPFL and GIS, JPF-Ducret SA and André SA, as well as the Bureau d'études Weinand and Yves Weinand Architectes SARL for the technological transfer.

References

[1] A. Stitic, C. Robeller and Y. Weinand, "Experimental investigation of the influence of integral mechanical attachments on structural behaviour of timber folded surface structures," *Thin-Walled Structures*, vol. 122, pp. 314-328, 2018.

[2] A. Stitic, A. C. Nguyen, A. Rezaei Rad, and Y. Weinand, "Numerical Simulation of the Semi-Rigid Behaviour of Integrally Attached Timber Folded Surface Structures," *Buildings, Multidisciplinary Digital Publishing Institute*, vol. 9, issue 2, pp. 5, 2019.

[3] A. C. Nguyen, and Y. Weinand, "Development of a spring model for the structural analysis of a double-layered timber plate structure with through-tenon joints", *WCTE 2018 - World Conference on Timber Engineering; South Korea; 20-23 August 2018;*

[4] K. Pukanska, J. Gajdosk, M. Marcian and K. Bartos "Determination of Deformations of a High-Capacity Tank Using the Technology of Terrestrial Laser Scanning", *American International Journal of Contemporary Research*, Vol. 4, No. 4; April 2014.

[5] J-A. Paffenholz and D. Wujanz "Spatio-temporal monitoring of a bridge based on 3D point clouds: A comparison among several deformation measurement approaches", *JISDM - Joint International Symposium on Deformation Monitoring, 15-17 May 2019, Athens, Greece.*