# TIMBER INTERACTIVE MODELLING AND POST-PROCESSING SOFTWARE (TIMPS)

**Manuel Sánchez-Solís[1], Màxim Lliso[2], Fernando Ibáñez[3]**

**ABSTRACT:** Timber Interactive Modelling and Post-Processing Software (TIMPS) is an internal programming routine implemented by Degree of Freedom for the scope of the analysis and design of timber structures acting as a plug-in for the most common commercial FEM software. Design timber codes and technical approvals from manufacturers are continuously being updated, therefore, design methods should also be adjusted to the latest equations. TIMPS is able to be customized for each project according to the current regulations and previous experiences in order to optimize the design of glulam members, CLT panels and connections specially for constructions with large amount of timber structural elements. The mechanical properties of the timber and its connections are calculated and transformed into typical inputs of commercial software, where the analysis is performed. Once the analysis is finished, TIMPS reads the software output and performs all verifications included in the codes. When the structure is optimized, final reports and graphics are printed.

## 1 INTRODUCTION

In the last two decades, the use of wood as a structural material in construction has experienced exponential growth all around the world. In some countries, structural timber is usually used in roofs, walkways, and single-family homes, limiting its use in big buildings which tend to have additional complexity, with a large floor volume or even a considerable height (up to 20-25 floors).

In this last application, the structural typology based on surface elements (walls and slabs) predominates globally, using panels of cross-laminated wood or CLT.

Regarding the structural design of this type of buildings, two great challenges appear compared to the calculation of conventional buildings: the modelling/verification of the linear elements and panels as structural elements and the modelling/design of the joints between panels.

One of the main characteristics of CLT panels is their orthotropy (both sectional and at the material level), which adds complexity to structural modelling in non-specific commercial calculation software, since a pre-process is required to transform the characteristics of the panel. to the characteristics of the finite element (different for each software).

Regarding its verification, again a specific formulation is required, not included in the usual software, which requires an external post-processing by the user to transform the forces of the finite element into stresses and demand/capacity ratios.

In relation to the joints between panels, the situation is analogous. It is possible to model them in conventional software with additional effort, while external post-processing is required to allow the joints to be dimensioned.

Regarding the first point, there are several programs aimed at sectional calculation and design of isolated elements of timber. The use of these programs is very useful in the design of small-scale buildings and in the pre-dimensioning of elements.

Among the commercial calculation software, there is a range of alternatives for the design of both linear and planar timber elements, however, not always the demands from the project are satisfied because of the available manufacturers of each region or the latest technical approvals or codes.

Furthermore, this software does not resolve all aspects of the design, since the results in joints between panels are limited to the mere obtaining of forces, also requiring external post-processing. Lastly, they do not have the versatility and power of other more advanced commercial software that allow for more complex applications to be carried out.

## 2 MODELLING SPECIFICITIES USING FEM FOR WOODEN STRUCTURES

FEM is a powerful extended tool in the structural professional practice regarding common structural elements (beams, shells) and materials (steel, concrete) thanks to its calculation capacity which translates to both precision and speed. However, important mechanical concepts need to be taken into account for the modelling of CLT panels and the adaptation of some aspects must be

[1] Manuel Sánchez-Solís, Degree of Freedom SLU, Valencia, Spain, m.sanchez@dofengineers.com
[2] Màxim Lliso, Degree of Freedom SLU, Valencia, Spain, m.lliso@dofengineers.com
[3] Fernando Ibáñez, Degree of Freedom SLU, Valencia, Spain, f.ibanez@dofengineers.com

implemented in order to correctly represent and control the behaviour of this shell elements.
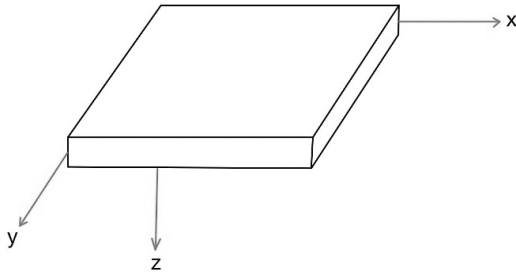


*Figure 1: Axes of orthotropic shells*

## 2.1 MATERIAL AND SECTIONAL ORTHOTROPIC PROPERTIES

CLT panels are shell elements constituted by different wooden layers which are orthogonally orientated with respect to the direction of their fibres. This fact consequently introduces two types of orthotropy: a material orthotropy in each layer and a sectional orthotropy in a given cross-section. In structural terms this has a particular impact on the shell stiffness since the double orthogonal symmetry of an orthotropic material originates a specific stiffness matrix shape.

In the case of isotropic shells in an elastic regime respecting the Infinitesimal Strain Theory, the stiffness matrix regarding stresses and strains is:

$$C = \begin{bmatrix} C_{11} & C_{12} & C_{13} & 0 & 0 & 0 \\ C_{21} & C_{22} & C_{23} & 0 & 0 & 0 \\ C_{31} & C_{32} & C_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & D_{11} & 0 & 0 \\ 0 & 0 & 0 & 0 & D_{22} & 0 \\ 0 & 0 & 0 & 0 & 0 & D_{33} \end{bmatrix} \quad (1)$$

In which the terms are:

$$C_{ii} = \frac{(1-v)E}{(1+v)(1-2v)} \quad (2)$$

$$C_{ij} = \frac{v\,E}{(1+v)(1-2v)} \quad (3)$$

$$D_{ii} = \frac{E}{2\,(1+v)} \quad (4)$$

Note the fact that only two parameters, the Young Modulus and the Poisson Coefficient, are required to define the stiffness matrix of an isotropic material.

As the study case is based on an orthotropic material, more parameters are needed to define the elastic properties of a cross-section. Those parameters are:
- Three Young Modulus:
  - $E_{11}$, $E_{22}$ and $E_{33}$
- Three Poisson Coefficients:
  - $v_{12}$, $v_{13}$ and $v_{23}$
- Three Shear Modulus:
  - $G_{12}$, $G_{13}$ and $G_{23}$

The stiffness matrix coefficients become then:
- Axial part:

$$C_{ii} = \frac{E_i(1 - v_{jk}v_{kj})}{\Delta} \quad (5)$$

$$C_{ij} = \frac{E_j(v_{ij} + v_{kj}v_{ik})}{\Delta} \quad (6)$$

- Transverse part:

$$D_{ii} = \Delta G_{jk} \quad (7)$$

Where indices $i, j$ and $k$ are always different and vary from 1 to 3 and:

$$\Delta = 1 - v_{23}v_{32} - v_{31}v_{13} - v_{12}v_{21} \\ - v_{12}v_{23}v_{31} - v_{32}v_{21}v_{13} \quad (8)$$

In the context of plate mechanics, no stress or strain is considered in the Z axe and a planar stress state occurs. Furthermore, it is possible to work with only one stiffness matrix with the shape of the one regarding the isotropic case if the cross-sectional orthotropy is physically represented in an equivalent single-layer shell. A method that fulfils this purpose is defined in section 3.1. By contrast, several coupled stiffness matrixes (one per layer) should be used to represent the global behaviour of the shell, which makes the analysis more complex.

These parameters are defined by the user in a later stage.

## 2.2 FINITE ELEMENT METHOD IMPLEMENTATION

As the FEM for static cases operates with force, stiffness, and displacement matrixes, these must be defined. The stiffness matrix of one layer is then computed in terms of force and displacement assuming the inexistence of work deformation in the Z axis (perpendicular to the plane of the shell). This matrix is introduced by the user into the calculation tool and will be called the elementary stiffness matrix hereafter.

Thus, external forces are:

$$F = \begin{pmatrix} M_x \\ M_y \\ M_{x,y} \\ V_x \\ V_y \\ N_x \\ N_y \\ N_{x,y} \end{pmatrix} \quad (9)$$

Where $m_x$ and $m_y$ are bending moments, $m_{x,y}$ is the torsional torque, $V_x$ and $V_y$ are the shear efforts due to bending, $n_x$ and $n_y$ are the membrane efforts and finally $n_{x,y}$ is the shear effort due to membrane loading. The associated stiffness matrix is:

$$K = \begin{bmatrix} k_x & k_v & 0 & 0 & 0 & 0 & 0 & 0 \\ k_v & k_y & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & k_{x,y} & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & S_x & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & S_y & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & D_x & D_{x,y} & 0 \\ 0 & 0 & 0 & 0 & 0 & D_{x,y} & D_y & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & D_S \end{bmatrix} \quad (10)$$

These terms are calculated according to the Eurocode 5: Design of Timber Structures and Reissner-Mindlin plate theory, where $K_z$ bending is not considered.
Finally, the displacement vector is given as:

$$U = \begin{pmatrix} \dfrac{\partial \Phi y}{\partial x} \\ -\dfrac{\partial \Phi x}{\partial y} \\ \dfrac{\partial \Phi y}{\partial y} - \dfrac{\partial \Phi x}{\partial x} \\ \dfrac{\partial uz}{\partial x} + \Phi y \\ \dfrac{\partial uz}{\partial y} - \Phi x \\ \dfrac{\partial ux}{\partial x} \\ \dfrac{\partial uy}{\partial y} \\ \dfrac{\partial ux}{\partial y} - \dfrac{\partial uy}{\partial x} \end{pmatrix} \quad (11)$$

Obviously, this formulation applies in the Continuous Medium Mechanics framework. The conversion into finite element exploitable objects is automatically done by the industrial calculation software by meshing the shells. Consequently, the size of the finite element stiffness matrix will depend on the level of discretisation reached. All equations above are further simplified for frame finite elements (beams and columns).

### 2.3 STRUCTURAL JOINTS

The connection between different elements is modelled by the use of the finite element type "link" from industrial software, which connects two nodes with different properties. The stiffness for each degree of freedom is defined according to the slip modulus calculated for each connection.
For direct contact connections, a high stiffness value is defined to ensure that the loads are transferred as expected. For released degrees of freedom, a low stiffness value is used (instead of 0) to prevent from numerical errors during analysis.
The slip modulus to define the correct stiffness value of the connection is calculated according to Eurocode 5, chapter 7.1. The table below show the equations to obtain the shear stiffness of the connections based on the connector type, timber density and diameter of the screws. The axial slip modulus is not included in the Eurocode as these values are normally defined by the producers.

**Table 1:** *Values of $k_{ser}$ for fasteners and connectors in N/mm in timber-to-timber and wook-based panel-to-timber connections. EC 5.*

| Fastener type | $K_{ser}$ |
|---|---|
| Dowels | |
| Bolts with or without clearance[a] | $\rho_m^{1,5} \cdot d/23$ |
| Screws | |
| Nails (with pre-drilling) | |
| Nails (without pre-drilling) | $\rho_m^{1,5} \cdot d^{0,8}/30$ |
| Staples | $\rho_m^{1,5} \cdot d^{0,8}/80$ |
| Split-ring connectors A (EN 912) | $\rho_m \cdot d_c/2$ |
| Shear-plate connectors B (EN 912) | |
| Toothed-plate connectors: | |
| - C1 to C9 (EN 912) | $1,5 \cdot \rho_m \cdot d_c/4$ |
| - C10 and C11 (EN 912) | $\rho_m \cdot d_c/2$ |

[a]The clearance should be added separately to the deformation

Th Eurocode states that for steel-to-timber connections, shear stiffness can be multiplied by 2.
The way in which screws are oriented along a CLT joint has an impact on both strength and stiffness of the connection. It is commonly assumed in the timber design references that only screws inserted perpendicular to the external force will work with a shear mechanism while any inclination of the fastener will make it work axially. Therefore, crossed-screws connections are very typical in timber industry. This concept is described in the following figure.
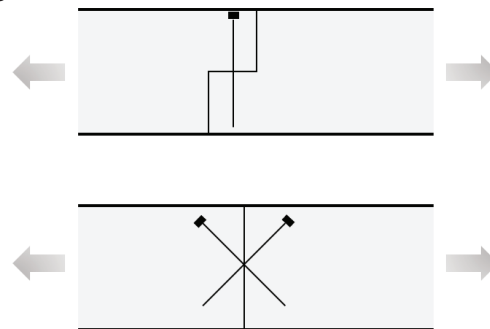


**Figure 2:** *Different screws resistance mechanisms*

## 3 TIMPS WORKFLOW

The improvement achieved in the logistical aspect of data processing is obvious. All the design process is now confined in two interfaces (the industrial FEM software and our self-elaborated routine) and the need of manually importing data sets is removed. The calculation stages are explained hereafter.

### 3.1 COMPUTATION OF THE ELEMENTARY STIFFNESS MATRIX

The elementary stiffness matrix is computed by using the Gamma Method for wooden structures developed in the Eurocode 5 or the Extended Gamma Method if the cross-section contains more than three longitudinal layers. By using this method, the shear stiffness of each layer is considered in the calculus by diminishing the second moment of inertia of the whole cross-section. This

approximation allows for the transition from a multi-layered material to a single-layered one, making it possible to build a single stiffness matrix for the complete orthotropic cross-section. Moreover, the coefficients of wood's transverse expansion $\nu_{ij}$ are often considered to be zero. This results in a diagonal matrix since $k_\nu = D_{x,y} = 0$ and therefore in a classical beam stiffness matrix for a generalised 3D situation.

## 3.2 STRUCTURAL MODELLING AND COMPUTATION

Structural modelling and computation are the only parts of the process elaborated by means of industrial FEM software. Therefore, the user defines the geometry, its boundary conditions, the loads and the load combinations of a given structure using wooden beams and CLT shells, which will be identified by groups. These shells are meshed and the model is computed. Various files containing the computation results are finally automatically created. Among data results, those of interest are the ones associated to nodal forces since they are needed in order to perform the stress analysis and verifications at the multi-layered element scale.

## 3.3 MODEL DATA ACCES THROUGH PROGRAMMING TOOLS

Once the FEM model is computed, several data packages are extracted using self-elaborated Python routines. Then, results can be selected, exported and manipulated. A significant improvement regarding post-processing computations is achieved by filtering datasets because the amount of data to be treated is well classified and defined. This improvement is appreciated when working with wooden buildings of more than ten floors since more structural effects and parameters, particularly linked to wind loading, need to be precisely considered. This translates into a relatively large amount of data to manage, so it is essential to only work with the necessary inputs with the aim of making post-processing approachable. The extracted data is then:
- Connectivity chart: this chart identifies each finite element with the nodes that form it.
- Node properties: each node is identified with a number according to their group. Their degrees of freedom as well as their global coordinates are obtained.
- Nodal forces: forces at each node.

This data is stocked in text files.

## 3.4 MODEL DATA SELECTION BY STRUCTURAL GROUPS AND LOAD CASES/COMBINATIONS

As all data coming from the finite element model belongs to a certain node, it is possible to filter it by the group to which it belongs. As stated before, these groups are created by the user when modelling the structure by means of the industrial software. The advantage of working with group data is that it allows for the possibility to only analyse the selected CLT shells or glulam frames.

On the other hand, a load combination analysis is also feasible. Results of a certain load case or combination can be extracted as well as the results of all ULS or SLS combinations. It is also possible to determine the most unfavourable load combination for each structural element in ULS and SLS so the design calculus can be directly done.

Thus, not all the finite element model's generated data is used to further develop the post-processing analysis.

## 3.5 IMPLEMENTATION OF VERIFICATION EQUATIONS ACCORDING TO EUROPEAN STRUCTURAL REGULATION

Stress limits are stablished by the Eurocode 5: Design of Timber Structures for ULS. Deformation limits are stablished the same way for SLS.

These equations are implemented in the calculation tool. The ULS verifications carried out are listed below.

Axial and bending verification for timber elements:
- Tension:

$$\frac{\sigma_{t,0,d}}{f_{t,0,d}} + \frac{\sigma_{m,y,d}}{f_{m,y,d}} \leq 1 \qquad (12)$$

- Compression:

$$\left(\frac{\sigma_{c,0,d}}{f_{c,0,d}}\right)^2 + \frac{\sigma_{m,y,d}}{f_{m,y,d}} \leq 1 \qquad (13)$$

Axial and bending verification for walls and columns. Buckling:

$$\frac{\sigma_{c,0,d}}{k_{c,y}\,f_{c,0,d}} + \frac{\sigma_{m,d}}{f_{m,d}} \qquad (14)$$

Shear:

$$\tau_{v,R,d} \leq f_{v,R,d} \qquad (15)$$

In-plane shear for CLT panels:
- Shearing-off failure of the boards along a joint:
$$\tau_{v,s,d} \leq f_{v,CLT,d} \qquad (16)$$
- Shearing failure of the glued surfaces in the intersection points:
$$\tau_{v,T,d} \leq f_{T,CLT,d} \qquad (17)$$
- Shear failure of the entire plate:
$$\tau_{v,d} \leq f_{vgross,CLT,d} \qquad (18)$$

With regards to SLS verifications, quasi-permanent and characteristic design situations are taken into account. The verifications are the following:

Deflection:
- Characteristic design situation:
$$w_{inst} \leq \frac{L}{300} \qquad (19)$$
- Quasi-permanent design situation:
$$w_{fin} = w_{inst} + w_{creep} \leq \frac{L}{250} \qquad (20)$$

Shear wall deformation:

$$w_{hor} = w_M + w_V + w_z + 2w_F \leq \frac{H}{300} \qquad (21)$$

Where $w_M$ is the bending deformation, $w_V$ is the shear deformation, $w_z$ is the expansion of the tie rods and $w_F$ is the displacement of the joints between wall and ceiling.

## 3.6 RESULT PLOTTING

Designing structures is an iterative process in which many parameters need to be considered. The variation of one of those parameters can significantly impact the results and it may not be easy to detect them, so a graphical representation is needed. Nodal displacements and stresses of the CLT shells or timber frames are plotted so that structural design judgement can be properly done.

As the TIMPS can work with several industrial software by interacting with their API, an interpolation module is provided for the cases in which data extracted is not interpolated so accurate plots can be done.

### 3.6.1 Examples

This section includes some of the outputs provided by this tool in different projects.

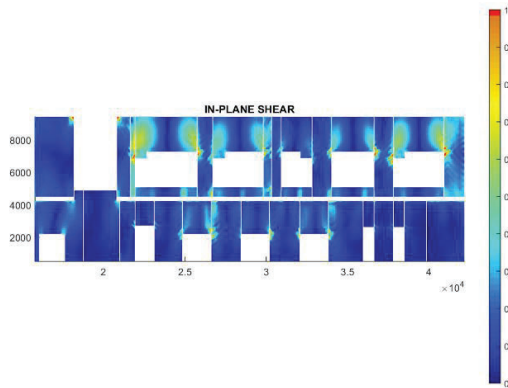Results for planar elements, linear elements and joints between CLT panels are shown below.



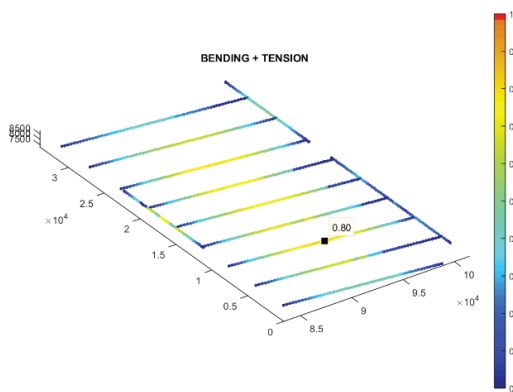**Figure 3:** *In-plane shear verification of façade CLT walls*



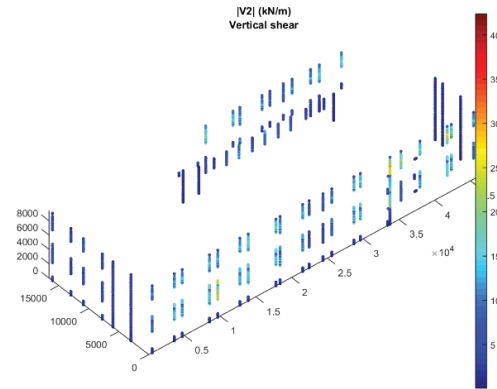**Figure 4:** *Bending verification of roof timber beams*



**Figure 5:** *Vertical shear forces in CLT joints*

### 3.6.2 Documentation generation

Once the iterative design process is finished, design reports and CAD files with required spacing of screws are exported.

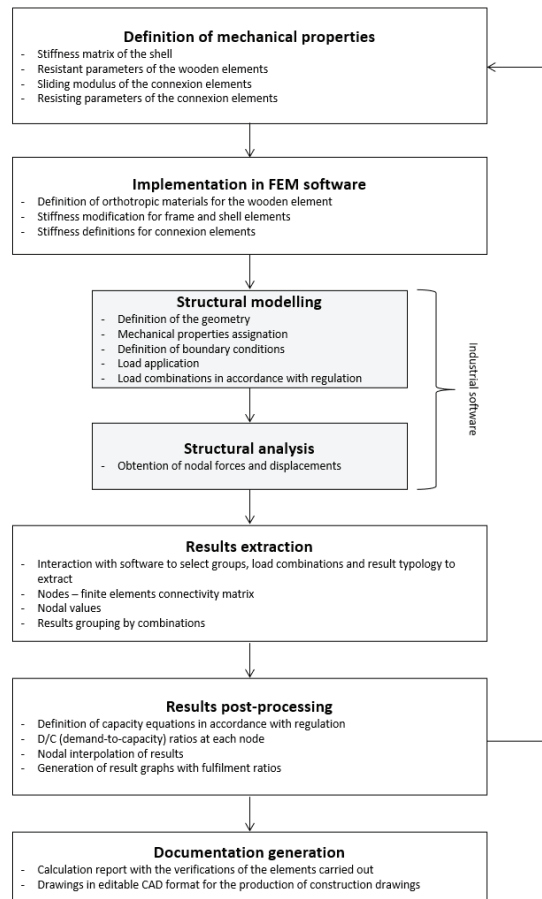The complete process performed by TIMPS is described in the following figure.



**Figure 6:** *TIMPS design workflow*

## 4 POTENTIAL IMPROVEMENTS

Several improvements are planned for the future of this tool. Some of them are listed below.

1) Predefinition of connection type catalogue with already calculated stiffness and strength to allow the user to check the most common joint solutions.
2) Connection to different software in the scope of the parametric design.
3) Improvement of CAD export function to minimize time consumed during detailed drawings production.
4) Evaluation of the transverse Poisson's Coefficients of the material in CLT shell computations. By considering them the results would be more accurate since the stiffness would be increased and best represented.

## 5 CONCLUSION

Timber Interactive Modelling and Post-Processing Software (TIMPS) is a design tool compatible with many of the commercial software for the purpose of the analysis, design and optimization of big timber structures based on the latest codes and technical approvals. The main goal of the application is to allow for quick updates of equations based on latest regulations and approaches from suppliers and producers in order to optimize future constructions by reducing the engineering time consumed for the verifications as a result of the iterative process inherent in the timber design sequence and the amount of data which needs to be evaluated by the structural engineer.

## REFERENCES

[1] Eurocode 5: Design of Timber Structures EN 1995.
[2] M. Wallner-Novak, J. Koppelhuber, K. Pock. Cross-Laminated Timber Structural Design. Basic design and engineering principles according to Eurocode, proHolz. Graz, Austria, 2014.
[3] M. Wallner-Novak, M. Augustin, J. Koppelhuber, K. Pock. Cross-Laminated Timber Structural Design. Volume 2, proHolz. Graz, Austria, 2018.
[4] Lebée, A. Modélisation des Structures Multicouches. Paris, France, 2022.