# Towards Sustainable Buildings: Novel Strategies for the Design of Vibration Resistant Cross-Laminated Timber Floors

WP3 – Numerical simulations and metamodeling

# Report on Upgraded Computational Tools

This research was supported by
the Science Fund of the Republic of Serbia, GRANT No 7677448

## Document Information

| | | | |
|---|---|---|---|
| **Grant Agreement Number** | 7677448 | **Acronym** | Substrate4CLT |
| **Full Title** | Towards Sustainable Buildings: Novel Strategies for the Design of Vibration Resistant Cross-Laminated Timber Floors | | |
| **Start Date** | 17th January 2022 | **Duration** | 36 months |
| **Project URL** | www.substrate4clt.com | | |
| **Deliverable** | D3.1 Report on upgraded computational tools | | |
| **Work Package** | WP3 – Numerical simulations and metamodeling | | |
| **Date of Delivery** | **Contractual** November 17th 2022 | **Actual** | November 17th 2022 |
| **Nature** | Report | **Dissemination Level** | Public |
| **Lead Beneficiary** | GRF-BG | | |
| **Responsible Author** | Miroslav Marjanović | | |
| **Contributions from** | Marija Milojević Emilija Jočić | | |

### Document History

| Version | Issue Date | Stage | Description | Contributor |
|---|---|---|---|---|
| V1 | November 13th 2022 | | | M. Marjanović, M. Milojević, E. Jočić |
| | | | | |
| | | | | |
| | | | | |
| | | | | |

# 1. Summary

The development of computational software tools and utilities is one of the most important activities of the Substrate4CLT project. As indicated in the project proposal, several team members have already been dealing with cutting-edge simulation tools based on dynamic stiffness and finite element methods in conjunction with different laminated plate and beam theories. Before the project started, some methods were implemented by the team members within the original computational tools [1, 2]. This served as a basis for the development of new tools and utilities within the Substrate4CLT.

The development of computational tools falls within the scope of WP3 - Numerical simulations and metamodeling. Within the WP3, computational tools needed for the vibration serviceability assessment of CLT floors have been developed, validated against the existing data from the literature, and designed to be easy for everyday use. After the initial setup of necessary tools, their development and calibration will be continued during the project implementation.

The development of computational tools and utilities is divided in the following groups, based on their purpose:

- calculation of mechanical properties of inter-panel connections in CLT,
- multi-modal analysis of CLT floors under human-induced loading,
- analysis of the progressive failure of CLT.

In the following sections, all the above aspects will be elaborated in detail.

# 2. Calculation of mechanical properties of inter-panel connections

Long-span cross-laminated timber (CLT) floors are typically an assembly of prefabricated CLT panels connected together on the site. Single panels are typically up to 3 m wide and less than 20 m long. These connections transfer in-plane shear forces and bending moments. The actual connections are commonly neglected in design calculations. Hence, a CLT floor is modelled either as a monolith slab or more frequently as a set of CLT panels with no connections. Four often-used connection types are illustrated in Figure 1.

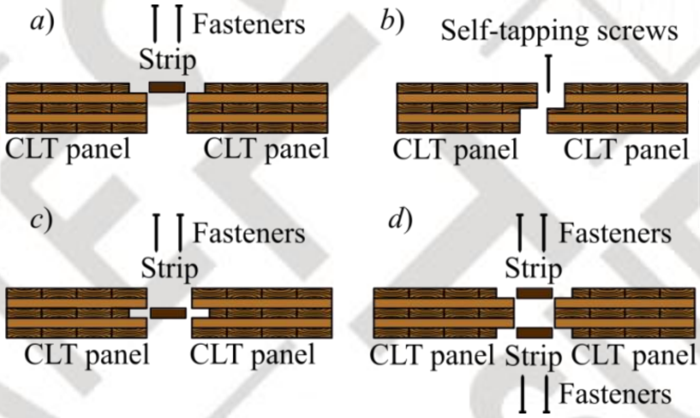


Figure 1: Four types of CLT inter-panel connections: a) single surface spline; b) half-lapped joint; c) internal spline, and d) double surface splines

One of the project goals is to prove that including the inter-panel connections significantly influences the dynamic behavior of CLT floors due to pedestrian-induced loading. Therefore, computational tools and utilities have been developed in order to calculate the mechanical properties of two commonly used connection types - single surface spline and half-lapped joint. The project also aims to provide the modeling strategy applicable in common design practice and thus uses a simple yet robust inter-panel connection model, regardless of finite element (FE) software used to extract vibration modes of a CLT floor. Computational models are extracted from [3, 4], as elaborated in the following two sub-sections.

## 2.1. Single surface spline

The single surface spline connection is modelled as an isotropic elastic strip having the width $a_{eq}$ and linked rigidly to the panels, as illustrated in Figure 2b. The elastic properties of the strip ($E_{eq}$ and $G_{eq}$) are derived according to [3], i.e. the real connection and the strip have the same rotation in a case of a constant moment $M$:

$$\phi_{eq} = \phi_{con}$$

$$\phi_{eq} = M\frac{l_{eq}}{E_{eq}I_{eq}}, \quad \phi_{con} = M\frac{l_{con}}{E_{con}I_{con}} \quad \rightarrow \quad E_{eq} = E_{con}\frac{l_{eq}I_{con}}{l_{con}I_{eq}} \tag{1}$$

where $\phi_{eq}$ and $\phi_{con}$ are the rotations of the equivalent elastic strip and the real connection, respectively, $l_i$ are the lengths of the cantilever, while $E_iI_i$ are the corresponding bending stiffnesses.

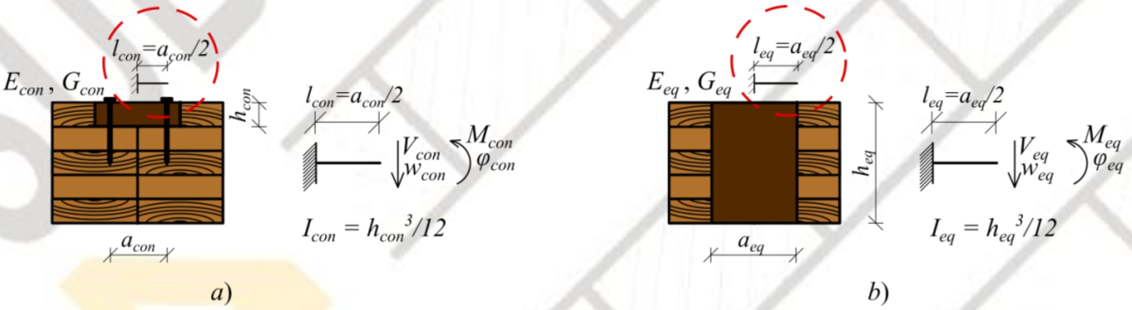


Figure 2: Single surface spline connection: a) actual, b) equivalent elastic strip

The shear modulus of the equivalent elastic strip, $G_{eq}$, is derived by matching the vertical displacements $w_{eq}$ and $w_{con}$ in the case of the constant shear force $V$:

$$w_{eq} = V\frac{l_{eq}^4}{3E_{eq}I_{eq}} + \frac{l_{eq}^4}{kG_{eq}A_{eq}}, \quad w_{con} = V\frac{l_{con}^4}{3E_{con}I_{con}} + \frac{l_{con}^4}{kG_{con}A_{con}}, \quad k = \frac{5}{6} \tag{2}$$

where $w_{eq}$ and $w_{con}$ are the vertical displacements of the equivalent elastic strip and the real connection, respectively, $k$ is the shear correction factor, while $G_iA_i$ are the corresponding shear stiffnesses.

| | A | B | C | D | E | F | G | H | I | J | K | L | M |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | Sve se računa od gornje ivice, i numeracija i dužine | | | | | | | | | | | | |
| 2 | b = 1m | | | | | | | | | | | | |
| 3 | i | h | $E_{long}$ | $G_{vert}$ | $z_s$ | $E_{long}\ h_i\ z_{s,i}$ | $E_{long}\ h_i$ | exc | $I_i$ | $E_{long,i}\ I_i$ | | | $G_{vert}$ A,i |
| 4 | - | mm | $N/mm^2$ | $N/mm^2$ | mm | N | N/mm | mm | $mm^4$ | $N\ mm^2$ | | | N |
| 5 | 1 | 30 | 370 | 50 | 15 | 166500 | 11100 | 60.00 | 110250000 | 40792500000 | | | 1500000 |
| 6 | 2 | 30 | 11000 | 690 | 45 | 14850000 | 330000 | 30.00 | 29250000 | 321750000000 | | | 20700000 |
| 7 | 3 | 30 | 370 | 50 | 75 | 832500 | 11100 | 0.00 | 2250000 | 832500000 | | | 1500000 |
| 8 | 4 | 30 | 11000 | 690 | 105 | 34650000 | 330000 | 30.00 | 29250000 | 321750000000 | | | 20700000 |
| 9 | 5 | 30 | 370 | 50 | 135 | 1498500 | 11100 | 60.00 | 110250000 | 40792500000 | | | 1500000 |
| 10 | | | $z_{board}$ [mm] = | 75 | | 51997500 | 693300 | | $EI_{board}$ $[kNm^2]$ = | 725.92 | | $GA_{board}$ [kN] = | 45900.00 |
| 11 | | | | | | | | | | | | | |
| 12 | | | | | | | | | | | | | |
| 13 | $a_{board}$ = | 100 | mm | | | | | | | | | | |
| 14 | | | | | | | | | | | | | |
| 15 | $\phi_{board}$ = | 0.001377567 | rad / kNm | | | | | | | | | | |
| 16 | | | | | | | | | | | | | |
| 17 | $w_{board,M}$ = | 4.59189E-07 | m / kN | | | | | | | | | | |
| 18 | $w_{board,V}$ = | 2.61438E-06 | m / kN | | | | | | | | | | |
| 19 | w = | 3.07357E-06 | m / kN | | | | | | | | | | |
| 20 | | | | | | | | | | | | | |
| 21 | Geometrija veze | | | | | Materijalne karakteristike veze | | | | | | | |
| 22 | $a_{con}$[mm]= | 100 | | | | | $E_{con}[kN/m^2]$= | 129052 | | 129052000 | N/m2 | | |
| 23 | $h_{con}$[mm]= | 150 | | | | | $G_{con}[kN/m^2]$= | 207728 | | 207727938.4 | N/m2 | | |
| 24 | $a_{con}/h_{con}$= | 0.6667 | | | | | | | | | | | |

Figure 3: MS Excel spreadsheet

The equivalent strip mechanical properties are calculated through the **ConnectionPropertiesSpreadsheet.xls** (Figure 3) developed by the project team.

## 2.2. Half-lapped joint

The connection is modelled according to [4] and involves calculating the stiffness of four springs arranged as illustrated in Figure 4b. The stiffnesses $k_{shear}$ and $k_{axial}$ correspond to the shear and axial stiffness of the screw, while $k_{top}$ and $k_{bottom}$ are related to the compression of the timber at the half-lap corners.

The relation between the moment M and the rotation θ is defined through the following relation [4]:

$$\frac{M}{\theta} = \frac{2h^2 k_{shear} k_{top}}{k_{shear} + k_{top}} + \frac{2l^2 k_{axial} k_{bottom}}{k_{axial} + k_{bottom}} \tag{3}$$

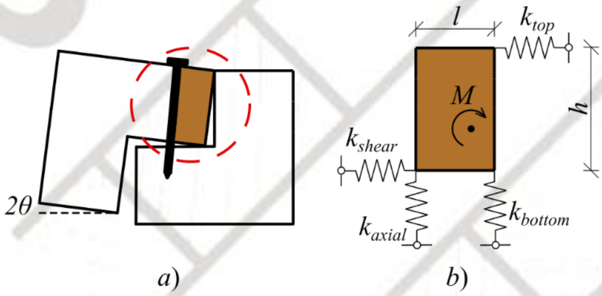


Figure 4. a) Real behaviour of a half-lapped joint, and b) rigid rectangle model (of length *l*) with four springs $k_i$ [4]

The equivalent strip mechanical properties are calculated through the **Rotational_stiffness_Macpherson.m** Matlab [5] script (Figure 5) developed by the project team.

Rotational_siffness_Macpherson.m

```
eq{4*brslojeva-1} = subs(M{brslojeva},x, A) == 0;
eq{4*brslojeva} = subs(Q{brslojeva},x,A) == 0;

S = vpasolve([eq{:}],C11, C12, C13, C14, C21, C22, C23, C24, C31, C32, C33, C34, C41, C42, C43, C44);

y{1} = subs(y{1}, [C11 C12 C13 C14], [double(S.C11) double(S.C12) double(S.C13) double(S.C14)]);

k_shear = abs(1000/double(subs(y{1},x,0)));

%% K TOP
if Lam(1) == lambda_l
    k_top = 5.0e3/0.2*a;%poduzna orijentacija vlakana
else
    k_top = 1.6e3/0.2*a; %poprecna orijentacija vlakana
end

%% K BOTTOM
if Lam(brslojeva) == lambda_l
    k_bottom = 3.3e3/0.2*a; %poduzna orijentacija vlakana
else
    k_bottom = 7.6e2/0.2*a; %poprecna orijentacija vlakana
end

%% K AXIAL

lambda = Eb/Gs;
zeta = log(5*(1-nu)*Lb/Db);
miL = Lb/Db*sqrt(8/zeta/lambda);
top = (4/(1-nu)+4*pi/zeta*tanh(miL)/miL*Lb/Db);
bottom = 1+1/pi/lambda*8/(1-nu)*tanh(miL)/miL*Lb/Db;

k_axial = abs(Gs*Db/2*top/bottom);
%% ROTATIONAL STIFFNESS

M = (2*h^2*k_shear*k_top/(k_shear+k_top)+2*l^2*k_axial*k_bottom/(k_axial+k_bottom))/a %u kNm/rad m'
Ma = M*a

fi = 1/M %rad/kNm
Econ = L/2/fi/I*1000 %N/m2
```

Figure 5. Rotational_stiffness_Macpherson.m Matlab script

# 3. HINDU computational tool

Within a project, a new computational tool named Hindu is developed to obtain the dynamic response of CLT floors due to human activities (walking or jumping). This Python-based [6] software is developed to use previously obtained dynamic (free vibration) properties of CLT floors.

After that, Hindu calculates the CLT floor's response by modal superposition technique and accounts for spatial variation of the human dynamic force. A unique numerical generator [7, 8] of artificial walking force signals at any pacing rate developed by Prof. Racić will be further used in simulations to calculate the dynamic response of investigated floors. By using such model, dynamic response can be obtained for a floor having an arbitrary fundamental frequency, i.e. regardless of the low or high frequency floor type.

Dynamic properties of simple floor layouts may be obtained using either [1] or commercial finite-element-method (FEM)-based software packages, while the properties of complex floor layouts (composed of several CLT panels, including joists, columns, etc.), as well as hybrid CLT-FRP, CONCRETE-CLT or CONCRETE-CLT45 floors should be performed using FEM. Within a project, software package Abaqus [9] will be used.

## 3.1. Basic principles

The response calculation process using Hindu software consists of several steps following the framework adopted in [10]. The first step is to define the floor's modal characteristics. As Hindu does not calculate the modal characteristics of the floor, they are imported from the Abaqus software. When the floor's characteristics are loaded, the next step is defining the *vibration source*, i.e. dynamic force generated by pedestrian walking. Generally, two types of floor response require different load models: one for resonant response (LFFs) and the other for transient response (HFFs). When assessing the vibration response of LFF, the walking force $F_p$ is assumed to be a perfectly periodic function, represented by a Fourier series:

$$F_p(t) = G + \sum_{h=1}^{N} \alpha_h G \sin(2\pi h f_p t - \varphi_h) \tag{4}$$

where: $G$ is the pedestrian weight; $\alpha_h$ is the Fourier's coefficient, or dynamic load factor (DLF) of the $h^{th}$ walking harmonic; $f_p$ is the walking frequency; $\varphi_h$ is the phase shift of the $h^{th}$ walking harmonic; $N$ is the total number of contributing harmonics.

Force models proposed by various authors differ according to the parameters used in Eq. (4), most commonly in the N and DLF values. Response of the HFFs is dominated by the impulsive, transient response. In that case, it is appropriate to model a dynamic load as a series of impulses representing each footstep.

The software user can choose between several implemented force models so far. These models are commercial and available in the literature [11-14]. Kerr [11] and Rainer [12] proposed harmonic force models based on Eq. (4); thus, they can be used when assessing only the response of the LFF. Arup [13] uses walking force models for both LFFs and HFFs. The methodology distinguishes if the floor is LFF or HFF based on the fundamental frequency and applies an appropriate dynamic force model. Živanović [14] suggested an advanced force model that can be used for both floor types. By choosing the start and endpoint, the user can set the straight walking path for moving dynamic force, simulating the pedestrian walking. After the *vibration receiver* point is defined, the vibration response can finally be calculated. As previously mentioned, the response calculation procedure is based on the modal superposition method. Therefore, it does not contain limitations regarding floor complexity or material.

## 3.2. Hindu graphical user interface

To make Hindu user-friendly, a graphical user interface (GUI) has been developed by using Python's package Tkinter as an application. It provides quick dynamic response calculation and effective visualisation of calculated responses.

Running the software opens the start window presented in Figure 6.

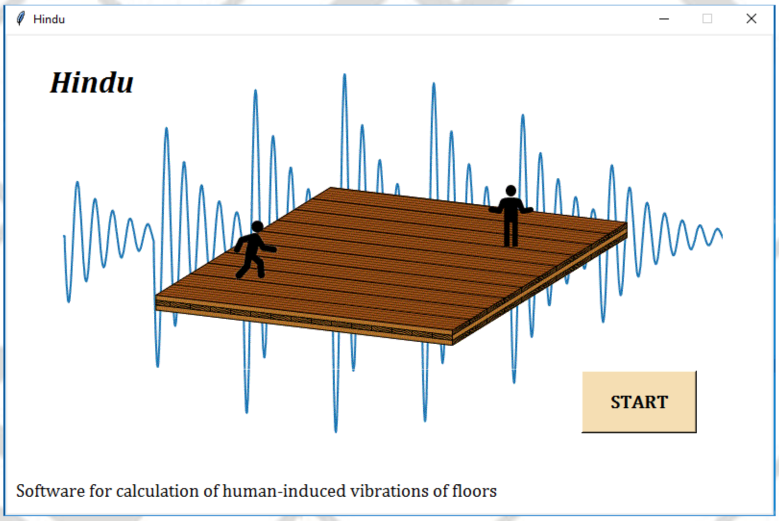


Figure 6. Hindu software start window

### 3.2.1. Main window

The start button on the start window opens the main window, where five critical sections of the main window are defined (Figure 7).

The Menu bar contains five tabs: File, Floor, Standards, Options and Help (Figure 7a). The tabs were designed to facilitate typical dynamic response analysis in Hindu.

The Mode selection section (Figure 7b) allows users to select modes to be included in the modal superposition-based dynamic response analysis.

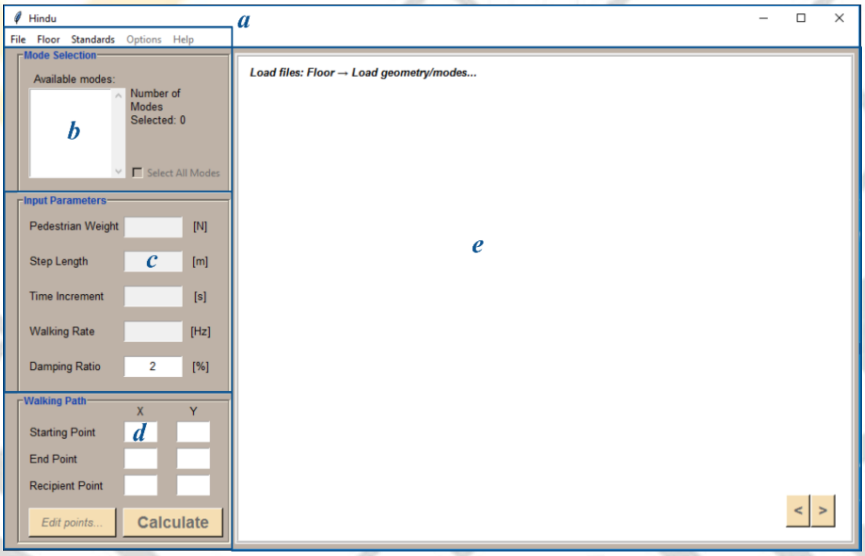


Figure 7. Hindu software main window and its components: a-Menu bar, b-Mode Selection, c-Input Parameters, d-Walking Path, e-Canvas

Before performing the analysis, several parameters should be defined in the Input Parameters section (Figure 7c). Dynamic load represented by the pedestrian weight, step length and walking frequency, as well as the damping ratio and time increment, are input parameters required to solve the SDOF equation of motion for each mode.

The user can define the walking path and the receiver point in the Walking Path section (Figure 7d).

Most of the main window is occupied by Canvas (Figure 7e), which displays and visualises the loaded floor's modal characteristics and results of the performed modal analysis.

## 3.2.2. Load FEM model utility

The Floor tab contains two options (Figure 8a): Load geometry/modes and Define walking path. Users can insert modal characteristics calculated in Abaqus by clicking the Load geometry/modes. When modal characteristics are loaded, they are graphically presented in Canvas (Figure 8b), and available modes are listed in section Mode Selection.

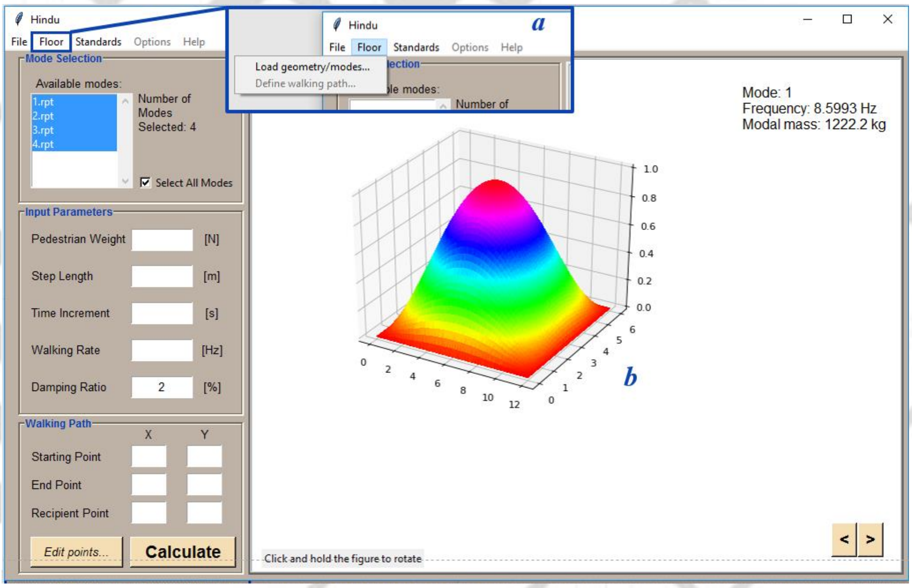


Figure 8. (a) Floor tab – available options, (b) Displayed loaded modal characteristics

## 3.2.3. Design guidelines and recommendations

The Standards tab enables users to select one of the above-discussed dynamic force models (Figure 9).

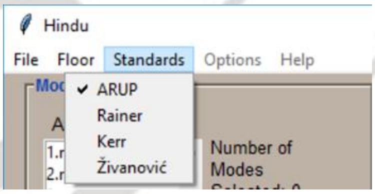


Figure 9. Standard tab – implemented procedures/dynamic force models

## 3.2.4. Visualisation & reports

Once all parameters are defined, the software calculates floor response in the receiver point by pressing the button Calculate. The time-history acceleration diagram is, by default, presented on Canvas, and the maximum value is given as well. In addition, the user can switch between the time-history diagrams for acceleration, velocity, displacement, or calculated so-called running arms trend (Figure 10).

Finally, the results of the performed analysis can be exported into .pdf and .doc file formats. The file contains the results of the modal analysis, specified input parameters, defined walking path, as well as time-history diagrams. Created reports can be attached to the floor design project.

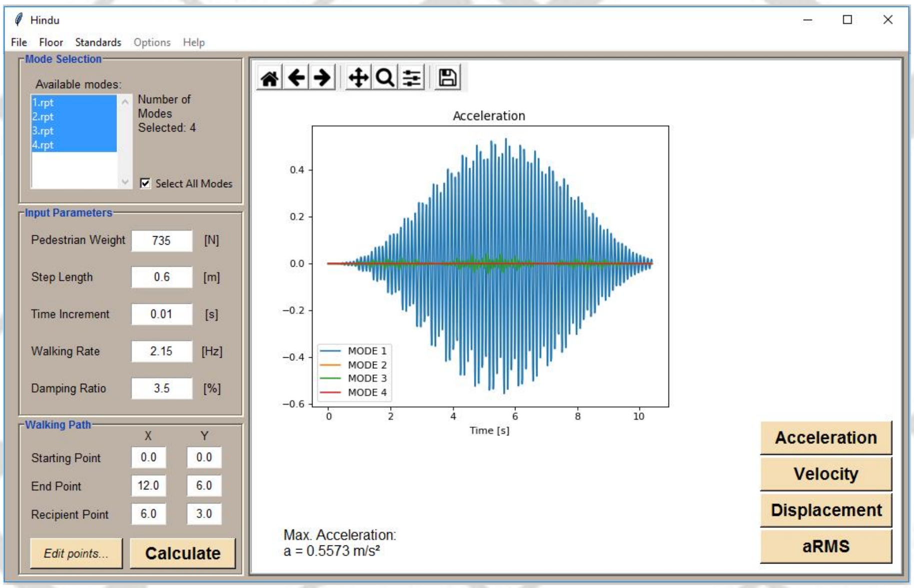


Figure 10. Displaying results on Canvas

### 3.2.5. Future work

The software has several limitations since it is still in its infancy. However, one of Hindu's essential features is the potential for further upgrades. The plan for future work will be directed toward creating possibilities to import modal properties from other FEM-based software (besides Abaqus CAE) or defining new modul so Hindu could be used to calculate the floor's dynamic properties.

The upgrade most relevant for the project would be implementing the dynamic force model developed by Prof. Racic. Afterwards, an appropriate probabilistic method will be adopted and implemented in the software to run probabilistic-based simulations.

# 4. Upgrade of the FLWTFEM software

The object-oriented computational framework FLWTFEM [2] is implemented in Matlab and shared with the user community through the GitHub repository (www.github.com/miregrf/FLWTFEM). It is primarily written for the analysis of laminated composite plates using layerwise plate theory [15], and implies the static and dynamic (free vibration) analysis of multilayered plate-like structures of arbitrary geometry. The pre- and post-processing phases are performed in conjunction with GiD software [16]. The FLWTFEM is being used and constantly improved by the authors for academic purposes.

The FLWTFEM solver is written using an object-oriented paradigm (see class structure in Figure 11), and implies the fast assembly procedure through the optimised (vectorised) algorithm for sparse matrices. In addition, a post-processing algorithm for calculating the accurate interlaminar stresses distribution through the plate thickness is implemented, satisfying continuity through the laminate thickness.

Within a Substrate4CLT project, FLWTFEM is extended by adding new analysis options:

- multi-modal analysis of multilayer plates, based on the previously calculated dynamic properties of multilayer plate-like structures,
- progressive failure analysis of multilayer structures, using the smeared-crack-band model.

The above improvements will be elaborated in the next sections.

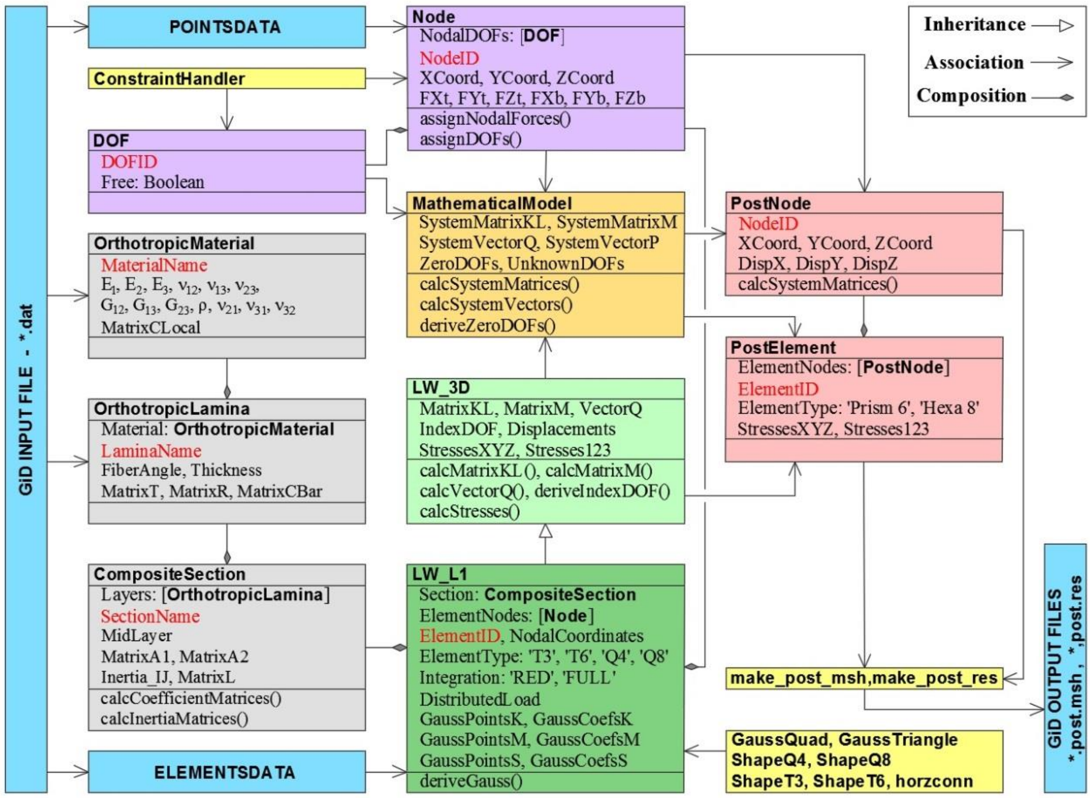


Figure 11. FLWTFEM class structure

## 4.1. Multi-Modal Analysis of CLT floors

By adopting the approach used in the development of Hindu (see section 3 for details), the upgrade of FLWTFEM for predicting the dynamic response of multilayer plates due to human activities has been done. The walking force model from [13] has been implemented.

As a general-purpose FEM-based code, FLWTFEM previously could predict the dynamic (free vibration) properties of multilayer plates, i.e. CLT floors, regardless of the plate geometry.

The FLWTFEM upgrade is done by editing the **FLWTFEM.m** Matlab script, and adding the following scripts: **walk_parameters.m**, **AppG.m**, **mode_resp.m**, **total_resp.m**, **mode_resp_all.m**, **response_all.m** and **max_acc_floor.m**.

Before performing the analysis, input parameters are defined: pedestrian weight, step length and walking frequency, damping ratio, time increment and total time, as well as the parameters (start and endpoint) of the straight walking path and recipient position. Once all parameters are defined, FLWTFEM performs the free vibration analysis of the considered plate structure and calculates floor response in the receiver point.

The previously developed conjunction between FLWTFEM and GiD [16] allows for adequate pre-processing, i.e. definition of geometrical entities (points, lines, surfaces or volumes), definition and assignment of attributes or conditions (BC, loads...) to geometrical entities, set up of analysis options and generation of the FE mesh.

The results are visualised in 3D by the GiD Post-Processing module. For visualisation purposes, FLWTFEM uses two classes (PostNode and PostElement), storing the necessary information for post-processing. The communication between the Matlab solver and GiD is accomplished using two output files: *.post.res and *.post.msh. The functions **make_post_res.m** and **make_post_msh.m** are modified within a project to generate the post-process files storing the information on plate displacements in all time steps during the simulation of pedestrian walking along the plate. The visualisation may also be made through an animation of plate motion, which may be saved in a .avi file.

The graphical user interface for pre- and post-processing in GiD is given in Figure 12.

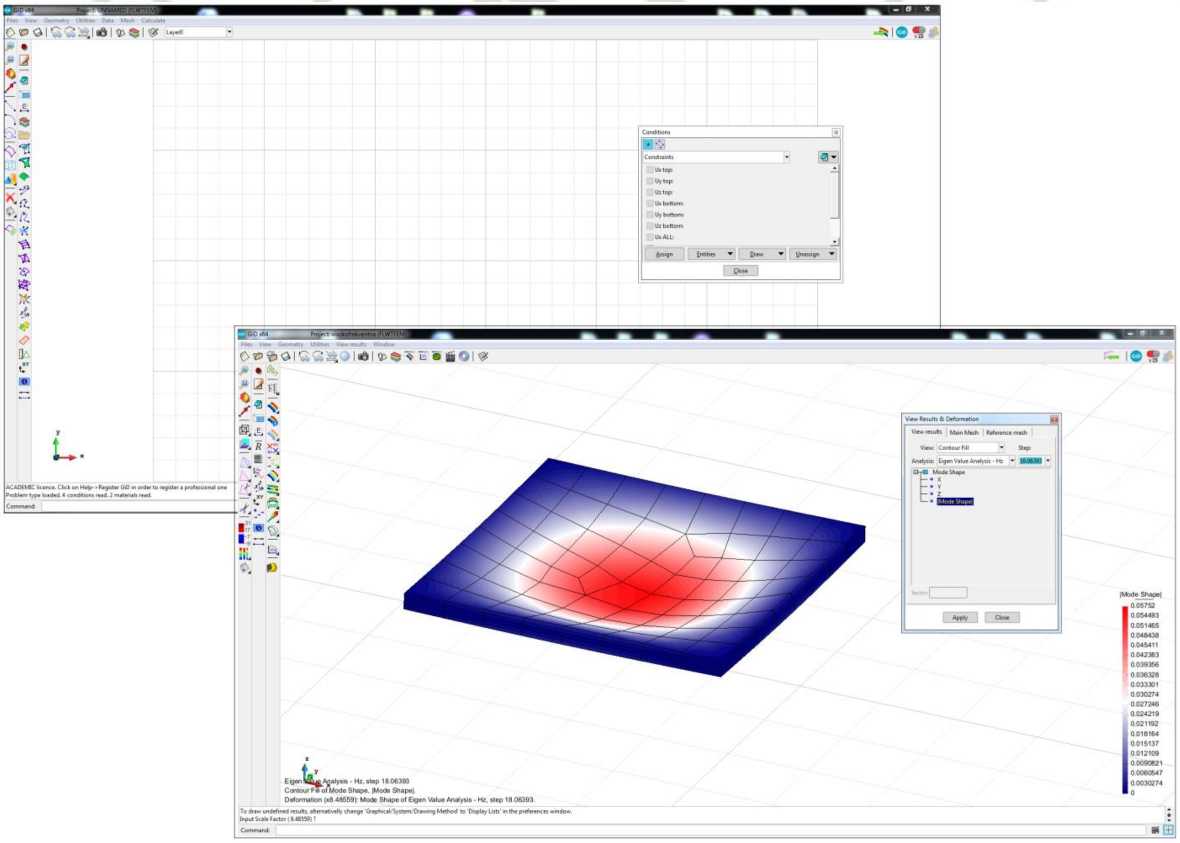

Figure 12. FLWTFEM GiD user interface for pre- and post-processing

## 4.2 Progressive failure analysis of CLT

In order to extend the knowledge on the progressive failure analysis (PFA) of cross-laminated timber (CLT) panels, the FLWTFEM was upgraded for prediction of post-failure behaviour of CLT due to damage progresion. The FLWT-SCB prediction model [17], previously developed by E. Jočić and M. Marjanović for PFA of laminar composites, has been modified and implemented.

In SCB approaches [18], the damage is smeared out within the finite element domain and the facture mechanism is then represented trough material stiffness degradation. The material stiffness matrix degradation was controlled by damage variables, which evolution are governed by an equivalent strains appropriately defined for each failure mode. The response of damaged lamina, in both fiber and matrix direction, was described by distinct bilinear strain-softening curves (see Figure 13), where the peak stress coincides with the fibre and matrix strength, respectively.

The mesh dependency problem was reduced by scaling the fracture energy using a characteristic element length ($l_c$), as described by the crack-band theory [19]. This damage law is determined based on the assumption that the total energy needed to fail an element (released strain energy) is equal to the energy needed to create a crack that passes through it (fracture toughness, $G$):

$$\varepsilon^f = \frac{2G}{\sigma_0 l_c} \tag{5}$$

In Eq. (5), $\varepsilon^f$ is maximum strain, while $\sigma_0$ is the material strength.

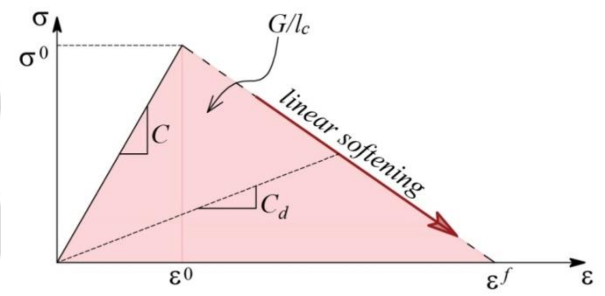


Figure 13. Stress-strain relationship with linear softening law

An object-oriented computational framework FLWTFEM already had the capacity to compute the 3D stress state of multilayer plates, i.e. CLT panels, regardless of the plate geometry, boundry conditions and stacking sequence. The FLWTFEM upgrade is done by editing the **FLWTFEM.m**, **LW_3D.m**, **OrthotropicMaterial.m** and **MathematicalModel.m** Matlab scripts.

Before performing the analysis, input parameters such as material strenghts, failure criteria, number of increments and iterations should be defined. By changing the **FLWTFEM.prb** input file in the GiD Pre-Processing module, the PFA was added in the AnalysisType dropdown menu (Figure 14) and the apropriate failure criterion can be selected. Also, **FLWTFEM.mat** input file was changed in order to add the material strengths within the existing mechanical properties of orthotropic material.

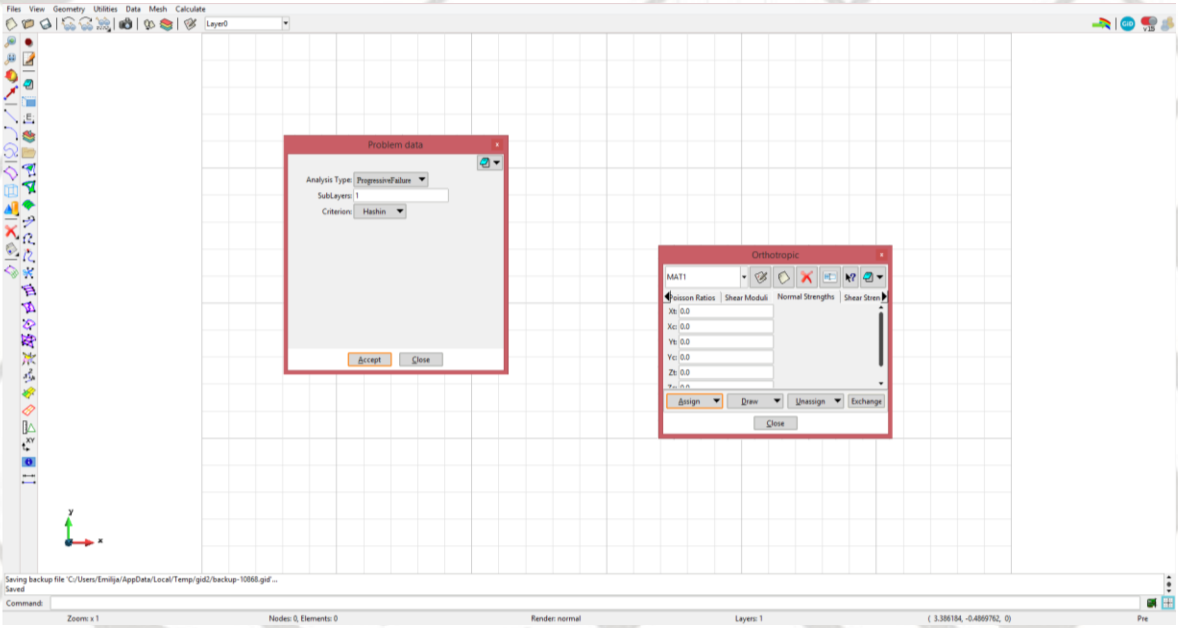

Figure 14. FLWTFEM GiD user interface for pre-processing

To produce the data input file to be processed by the solver within a simulation, GiD interprets the FLWTFEM.bas file, which describes the format and the structure of the required input data for the Matlab solver. Once the *.dat file is created, FLWTFEM performs the implicit PFA of the considered CLT panel, and then calculates the damage variables in each element of each lamina, in order do describe the response of damage laminas.

After the calculation is finished, the two output files: *.post.res and *.post.msh are created, and the results are visualized in 3D by the GiD Post-Processing module. The functions **make_post_res.m** and **make_post_msh.m** are modified within a project, to generate the post-process files storing the information on damage variables in all increments during PFA of corresponding CLT panel (Figure 15).

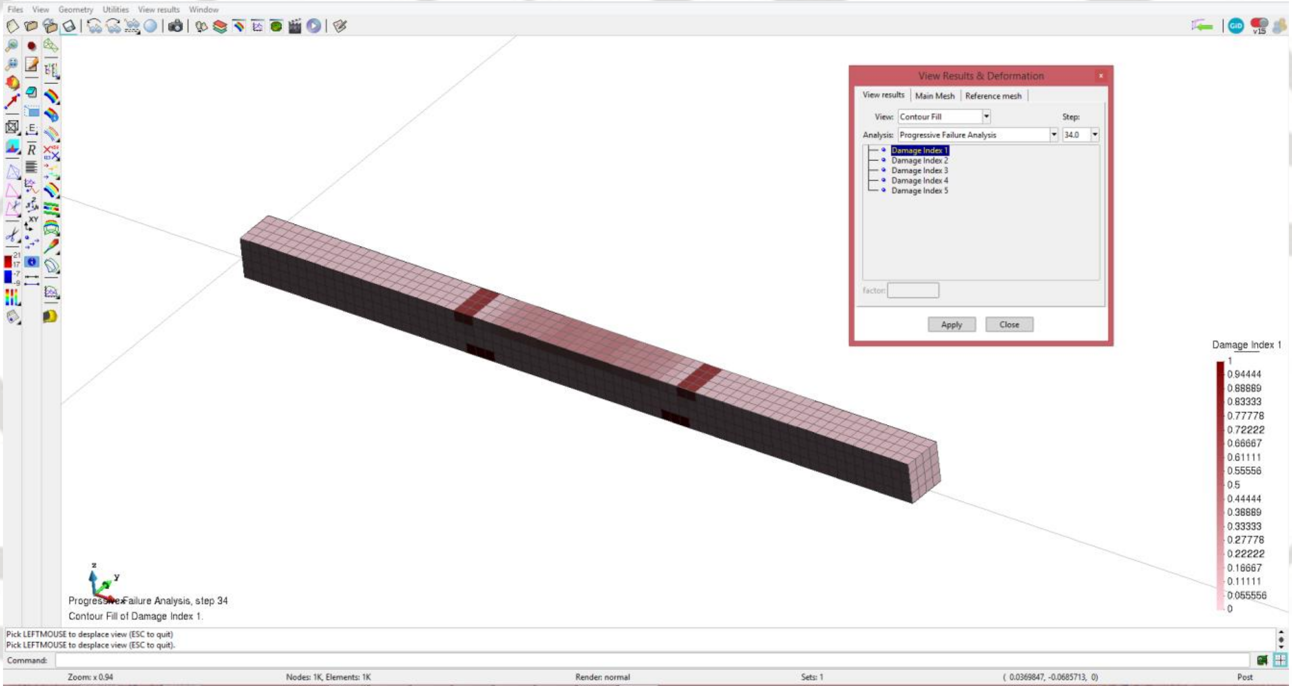


Figure 15. FLWTFEM GiD user interface for post-processing

# REFERENCES


[1] M. Marjanović, M. Nefovska-Danilović, E. Damnjanović (2019): Framework for dynamic-stiffness-based free vibration analysis of plate-like structures, Shock and Vibration, Paper 1369235.

[2] Marjanović M, Meschke G, Damnjanović E. Object-oriented framework for 3D bending and free vibration analysis of multilayer plates: Application to cross-laminated timber and soft-core sandwich panels. Composite Structures 2021; 255: Paper 112859.

[3] Paolini A, Kollmannsberger S, Winter C, Buchschmid M, Muller G, Rabold A, Mecking S, Schanda U, Rank E. A high order finite element model for vibration analysis of cross laminated timber assemblies. Building Acoustics 2017; 24(3): 135-158.

[4] Macpherson E, Papastavrou P, Wallwork T, Smith S, McRobie A. The rotational stiffness of cross-laminated timber half-lap joints. World Conference on Timber Engineering WCTE 2018. Seoul, Republic of Korea, 2018.

[5] Matlab. version 7.10.0 (R2010a). Natick, Massachusetts: The MathWorks Inc.; 2010.

[6] G. van Rossum: Python Tutorial 3.10.4, Python Software Foundation, 2022.

[7] V. Racić, J.M.W. Brownjohn (2011): Stochastic model of near-periodic vertical loads due to human walking, Advanced Engineering Informatics 25, 259-275

[8] V. Racić, A. Pavić (2009) Mathematical model to generate asymmetric pulses due to human jumping, ASCE Journal of Engineering Mechanics 135(10), 1206-1211

[9] ABAQUS User Manual 6.9. DS SIMULIA Corp., Providence, Rhode Island, USA, 2009.

[10] Bases for design of structures - Serviceability of buildings and walkways against vibrations, International Organization for Standardization, 2006.

[11] S. Kerr, N. Bishop. Human induced loading on flexible staircases, Engineering Structures, vol. 23, pp. 37-45, 2001.

[12] J. H. Rainer, G. Pernica, D. E. Allen. Dynamic loading and response of footbridges, Canadian Journal of Civil Engineering, vol. 15, pp. 66-71, 1988.

[13] M. Willford, P. Young. A Design Guide for Footfall Induced Vibration of Structures, The Concrete Centre, London, 2006.

[14] S. Živanović, A. Pavić. Probabilistic Modeling of Walking Excitation for Building Floors, Journal of Performance of Constructed Facilities, vol. 23, no. 3, pp. 132-143, 2009.

[15] Reddy JN. Mechanics of laminated composite plates and shells: Theory and analysis. 2nd edition, CRC Press, Boca Raton, Florida, 2003.

[16] Coll A, Ribó R, Pasenau M, Escolano E, Perez J, Melendo A, Monros A, Gárate J. GiD v.13 User Manual, CIMNE - International Centre for Numerical Methods in Engineering, Barcelona, 2016.

[17] Jočić E, Marjanović M. Progressive failure analysis of open-hole composite laminates using FLWT-SCB prediction model. International Journal of Mechanical Sciences 2022; 227: 107407.

[18] Forghani A, Poursartip A, Vaziri R. An orthotropic non-local approach to modeling intra-laminar damage progression in laminated composites. International Journal of Solids and Structures 2019; 180: 160-175.

[19] Bazant Z. P. Instability, ductility, and size effect in strain-softening concrete. Journal of Engineering Mechanics -Asce 1976; 102(2): 331-344.

# End of Document

This research was supported by
the Science Fund of the Republic of Serbia, GRANT No 7677448