© OCT 2023 | IRE Journals | Volume 7 Issue 4 | ISSN: 2456-8880

Finite Element Analysis of Moment Resisting Joint in Timber Frames

ABDULKADIR YUSUF
Department of Civil Engineering, Kampala International University, Uganda

Abstract- An important responsibility of a structural engineer is to develop suitable arrangement of structural system that can withstand the most critical combination of load cases efficiently during the entire lifetime of the structure. The basic deterministic finite element analysis based on displacement-based finite element method using commercial software Staad.pro was used. The versatility of the software is, its ability to model different materials, differing structural configuration, and also capable of performing linear, nonlinear, buckling and pushover analysis. The essential need is to have a tool capable of predicting the state of stress and strain in any element of the structural system. To verify the accuracy of the analysis, a 25m span pitched timber portal frame made from glued-laminated (glulam) material was discretized and all the loadings (dead, imposed, snow and wind loads) applied. Using eight nodesisoparimetric type solid elements to realistically model the entire frame, the equilibrium condition of the elements within the joint area was captured. Thus, the critical shear capacity required for the design of joint in the frame was determined to be 29.047KN.

Indexed Terms- Finite Element Method, Solid Elements, Timber Portal Frames.

I. INTRODUCTION

A fundamental requirement for effective utilization of wood as a competitive structural material is the accurate knowledge of the mechanical properties. However, the experimental identification and the analytical modeling of the mechanical behavior of wood remain an open problem, due to the natural variability, in homogeneity and anisotropy [1]. In the framework of synthetic composites an intensive research effort has been devoted to the experimental identification of the stress-strain relationships, including the ultimate stresses [2-8]. The same does not hold for solid wood, for which there are few studies about its orthotropic mechanical behavior [9-14]. The majority of published works devoted to this subject focused on the initial elastic behavior [15-17]. However, it is clear from the results published by several authors that wood possesses a nonlinear mechanical behavior [18]. Hence, a more realistic representation of mechanical behavior of wood is needed for some structural problems, like those in the mechanics of timber joints [17].

II. ACTIONS ON TIMBER FRAMES

For an intended construction work, the designer is first faced with the conceptual design of the structural system. This stage will consider the type of structure and the construction material to be used. The structural design then starts with an analysis of the actions that may be applied to the chosen structure. Account is taken of the direct actions that are the applied external forces as well as the indirect actions that are result from imposed deformations (e.g. settlement of supports or dimensional change induced by moisture variations).

Regardless of the construction material, the design requires the evaluation of actions that may act during the life of the structure. These depend on the structural form, on the type of construction work and the method of construction. At this stage, it is necessary to consider the nature of the actions or action effects, i.e. either static or dynamic, to achieve an accurate structural analysis. For example, the quasi-static assumption may not be acceptable in the following cases:

- floors subjected to human or machine-induced vibrations.
- Flexible plate-like structures such as suspension bridge decks that could flutter when subjected to wind velocities above a critical value,

- Structures loaded by ground acceleration due to seismic action.

In these cases, a dynamic analysis of the model is used to find the action effects of the force-time history, considering the stiffness, the mass and the damping ratio of the structural members. However, the resonant design methods component of action-effect is small for most structures. Therefore the static calculations are made, and an equivalent dynamic amplification factor applied to the static value of action.

The design Eurocodes (Eurocode-2 to Eurocode-7) are based on a calibration of successful traditional design methods. Nevertheless, a mention should be made of the criteria to which the reliability concept of Eurocode-1 referred. Regarding human hazard and economic losses, the structural safety and serviceability requirements consider the working life and design situations of the structure [19]. The design working life is as shown in Table 1.

Table 1: Design working life classification.

Class	Working life (years)	Example
1	1 to 5	Temporary structures
2	25	Replaceable structural element
3	50	Buildings and common structures
4	100	Bridges or engineering works

The working life corresponds to the period for which the structure is to be used for its intended purpose. Table 1 gives a classification of construction structures. The design situations refer to events that may occur during the working life of the structure. Therefore, the actions are evaluated for relevant design situations that are classified as:

- persistent situations related to the conditions of normal use,
- transient situations related to temporary conditions, e.g. during execution, accidental situations related to exceptional conditions like impact.

2.1 DESIGN VALUES OF ACTIONS ON STRUCTURES

The design actions may be different for different limit states. Firstly, the possible load cases are identified, i.e., compatible load arrangements, sets of deformations and imperfections. A load arrangement identifies the position, magnitude and direction of an action. Secondly, the actions are combined according to the following symbolic expression:

$$\sum \gamma_{G_d} G_{kj} \text{ "+" } \gamma_{Q,1} Q_{k,1} \text{ "+" } \sum \gamma_{Q,i} \psi_{0,i} Q_{k,i} \quad (1)$$

whereg are partial factors (load factors) for the action considered, taking account of: the possibility of unfavorable deviation of the actions, the possibility of inaccurate modelling of the actions and uncertainties in the assessment of the effects of actions. The representative values multiplied by the g-values, $g_G G_k$, $g_G Q_k$,$g_G y_o Q_k$ are called design actions. The principle is thus that one variable action with its characteristic value in turn is combined with the permanent actions and all other variable actions with their combination value $y_0 Q_k$.

Finally, the effects (S) of actions, for example, internal forces and moments ($F_{d,1}$), stresses ($F_{d,2}$), strains and displacements ($a_{d,1}$), are determined from the design values of the actions, geometrical data and, where relevant, material properties(X).

$S_d = S(F_{d,1}, F_{d,2},..a_{d,1}, a_{d,2},., X_d,\ldots)$ (2)

As a simplification it is permitted instead of equation 2 to use the more adverse of the following combinations.

- For limit state with only one variable action: $SgG_{k,j}$ "+" $1.5Q_{k,1}$
- For other cases: $SgG_{k,j}$ "+" $1.35SQ_{k,i}$

In Eurocode5 [20] the load combinations for serviceability limit state are given as:

$$\sum G_{k,j} \text{ "+" } Q_{k,1} \text{ "+" } \sum \psi_{1,i} Q_{k,i} \quad (3)$$

The design value X_d of a material property with the characteristic value X_k is defined as:

$X_d = k_{mod} X_k / g_M$ (4)

$g_M$is the partial safety factor for the material property.

k_{mod} is a modification factor taking into account the effect on the strength parameters of the duration of the actions and the moisture content.

The Eurocode5 values for g_M are given as:
Ultimate limit states
Timber and wood-based materials 1.3
Steel used in joints 1.1

Serviceability limit states 1.0

The factor k_{mod} depends on the Service Class to which the structure belongs and the Load-duration Class. There are three Service Classes denoted 1, 2 and 3. The classes 1 and 2 are characterized by the moisture content of the surrounding air. In Service Class 1 the average equilibrium moisture content will not exceed 12%; in Service Class 2 it will not exceed 20%. There is no limit for Service Class 3.

2.2 COMBINATION OF ACTIONS

After the estimation of the actions, the design requires the structural analysis as a result of the action effects. This stage involves the selection of realistic load arrangements for which the structure or the structural components are to be designed. Then, the design values result from the following combination of actions. Firstly, at the ultimate limit states, the combination for persistent or transient situation is:

$$\sum_{i}\gamma_{G,i}G_{k,i} + 1.5\,Q_{k,1} + \sum_{j=1}1.5\psi_{0,j}Q_{k,j} \quad (5)$$

where $\gamma_{G,i}$ is the partial factor for the permanent loads

$Q_{k,j}$ represents the dominant variable actions.

Secondly, the combination at the serviceability limit states depends on the action effect being checked considering both:

the characteristic combination:

$$\sum_{i}G_{k,i} + Q_{k,i} + \sum_{j=1}\psi_{0,j}Q_{k,j} \quad (6)$$

and the quasi-permanent combination:

$$\sum_{i}G_{k,i} + \sum_{j=1}\psi_{2,j}Q_{k,j} \quad (7)$$

For timber structures, the designer must pay special attention to finding out the critical load cases as they depend on the load-duration factors. At the ultimate limit states, the combination (1) is related to the use of the k_{mod} factor. For each combination including variable actions, the appropriate k_{mod} corresponds to the dominant action $Q_{k,i}$. At serviceability limit states, combination (2) applies to the calculation of instantaneous action in service. In addition, the combination (3) refers to the calculation of the long-term action effect using relevant factor k_{def} for the material and service class of the structure [21].

Considering the different limit states, the combination of the action is calculated for each critical load case. The designer's judgment could lead him/her to consider a few worse case of load arrangement. These are commonly:

- (dead + imposed) for floor members or (dead + snow) for roof members
- (dead + wind + snow $S_1/2$ or S_2) for the structure.

Uniformly distributed loads usually control the design of members, while unbalanced load cases can induce more critical effects for connections or in some framing systems (i.e. lattice structure).

Depending on the effect being checked, the combinations of actions for ultimate limit state are:

C1: $1.35(g_k + G_k)$ (8)
C2: $1.35(g_k + G_k) + 1.5(q_k \text{ or } Q_k)$ (9)
C3: $1.35(g_k + G_k) + 1.5S_{1,k}$ (10)
C4: $(g_k + G_k) + 1.5w_{i,k}$ (11)

furthermore, the combination of snow and wind actions also for ultimate limit state are:

C5: $1.35(g_k + G_k) + 1.5w_{i,k} + 1.5y_{0,s}[\frac{S_{I,k}}{2} \text{ or } S_{II,k}]$ (12)

C6: $1.35(g_k + G_k) + 1.5[\frac{S_{I,k}}{2} \text{ or } S_{II,k}] + 1.5y_{0,w}\, wi_{,k}$ (13)

where$y_{0,s}$ and $y_{0,s}$ are the combination factors associated with snow and wind.

In practice the design of frame depends on the design of moment-resisting joint. Staad.pro provides the platform for the analysis in which all the envisaged load types and combinations are adequately presented. The critical conditions of stresses and deformations also form part of the results obtainable using Staad.pro[22].

The combinations of actions for serviceability limit state are now considered. As snow is the main variable action, the instantaneous effects of the actions are calculated from the combinations:

C7: $(g_k + G_k) + S_{I,k}$ (14)

and

C8: $(g_k + G_k) + S_{II,k} + y_{0,w}w_{i,k} = (g_k + G_k) + S_{II,k} + 0.6\, w_{i,k}$ (15)

Depending on the shape and the span of the frame, the limitation for the horizontal deflection of the column is checked using either the combination C7 or C8. The combination gives the maximum value of the vertical deflection at the pitch of the frame. In addition, the calculation of the long-term effects such as creep deformations refers to the quasi-permanent combinations:

C9: $(g_k + G_k) + y_{2,s}S_{I,k} = (g_k + G_k) + 0.1\, S_{I,k}$ (16)

C10: $(g_k + G_k) + y_{2,s}S_{I,k} + y_{2,w}w_{i,k} = (g_k + G_k) + 0.1\, S_{II,k}$ (17)

To calculate the final deflections, it is therefore necessary to consider the combinations:

- (C7, C9) for vertical displacements,
- (C7, C9) or (C8, C10) which ever causes the greater horizontal displacement.

III. CONCEPT OF FINITE ELEMENT ANALYSIS

The following analysis facilities are available in STAAD:

a. Stiffness Analysis/Static Analysis
b. Second Order Static Analysis
(i) P-Delta
(ii) Non-Linear Analysis
(iii) Multi Linear Spring Support
(iv) Member/Spring Tension/Compression only
c. Dynamic Analysis
(i) Time History
(ii) Response Spectrum

Static analysis method is used to obtain forces and moments design. Analysis is done for the primary and combination loading conditions provided by the user. The user is allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations. Depending on the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the result combined with static analysis results.

The assumptions and formulations involved in these analyses are consistent with Small Displacement Theory. The nodal stiffness matrices of the individual finite elements are first computed and then transformed from the local element coordinate system to global coordinate system. Finally, the individual stiffnesses associated with each nodal point are systematically summed to obtain the total (global) stiffness matrix [K]. This square symmetric matrix has up to 6 equations per node. For a complete analysis of the structure, the necessary matrices are generated on the basis of the following assumptions:

- The structure is idealised into an assembly of beam, plate, solid, spring, and matrix type elements joined together at their vertices (nodes). The assemblage is loaded and reacted by concentrated loads acting at the nodes. These loads may be both forces and moments which may act in any specified direction.
- A beam element is a longitudinal structural element having a constant, doubly symmetric or linearly tapered cross section along its length. Beam elements may carry axial forces, shear and bending in two arbitrary perpendicular planes, and are also subjected to torsion.
- A plate element is a three or four noded flat element having constant thickness and orthotropic properties.
- A solid element is a four to eight node three dimensional element having uniform orthotropic properties.
- A general matrix element may be a spring or generated element matrices or assembled stiffness/mass matrices from an external source.
- Beams have 1 to 6 degrees of freedom (DOF) at each node; plates have 5 DOF; and solids have 3 DOF. Because of this mismatch in DOF, the

connection between solid elements and plates or beams needs special modeling.

- The solved internal element forces and the applied external loads acting on each node are in equilibrium except at supports.
- Two types of coordinate systems are used in the generation of the required matrices and are referred to as local and global systems. Local coordinate axes are assigned to each individual element and are oriented such that computing effort for element stiffness matrices are generalized and minimized. The nodal coordinate are by definition in the global coordinate axis system. The assembled forces and stiffness and the solved displacements are in this global coordinate system.

Loads may be applied in the form of distributed loads on the element surfaces or as concentrated loads at the nodes, element thermal gradients, pressures, inertia (self-weight) loads and imposed nodal displacements. Using standard finite element methods the loadings are assembled into nodal force vector. During a static analysis, this matrix equation is solved:

[K] . {d} = {P} (20)

where

[K] = the stiffness matrix.

{d} = the resulting nodal displacement vectors.

{P} = applied nodal force vectors.

3.1 FORMULAE FOR ELEMENT MATRICES [K] AND $\{R_E\}$

The derivation of finite element formulae is a straightforward procedure, which can be summarised as follows [23]. Displacements are taken as the dependent variables. The approach is known as displacement formulation which is equivalent to the minimisation of the total potential energy of the system in terms of a prescribed displacement field. The process is then equivalent to the well known Rayleigh-Ritz procedure. Therefore, the appropriate function for a Rayleigh-Ritz solution is Π_p, the expression for potential energy. In Rayleigh-Ritz method the admissible displacement field is selected, defined in a piecewise fashion so that displacements within any element are interpolated from nodal DOF of that element, and then Π_p is evaluated in terms of the nodal DOF. Using the principle of stationary potential energy, $d\Pi_p = 0$, from which algebraic equations to be solved for nodal DOF are obtain. In the course of this argument certain expressions as the element stiffness matrix [K] and the element load vector $\{r_e\}$ are generated. Details of this derivation are now described.

3.2 SOLID ELEMENTS APPLICATION USING STAAD

Solid elements enable the solution of structural problems involving general three dimensional stresses. There is a class of problems such as stress distribution in concrete dams, soil and rock strata where finite element analysis using solid elements provides a powerful tool. The solid element used in STAAD [23] shown in Fig. 1, is of eight nodedisoparametric type. These elements have three translational degrees-of-freedom per node.

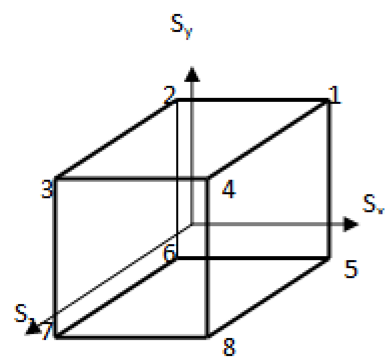

Fig. 1: An eight node solid

By collapsing various nodes together, an eight noded solid element can be degenerated to the other forms with four to seven nodes. The stiffness matrix of the solid element is evaluated by numerical integration with eight Gauss-Legendre points. To facilitate the numerical integration, the geometry of the element is expressed by interpolating functions using natural coordinate system, (r,s,t) of the element with its origin at the center of gravity. The interpolating functions are as follows:

$$x = \sum_{i=1}^{8} h_i x_i , \quad y = \sum_{i=1}^{8} h_i y_i , \quad z = \sum_{i=1}^{8} h_i z_i$$

wherex,y and z are the coordinates of any point in the element and x_i, y_i, z_i, i=1,..,8 are the coordinates of nodes defined in the global coordinate system. The interpolation functions, h_i are defined in the natural coordinate system, (r,s,t). Each of r,s and t varies between -1 and +1. The fundamental property of the unknown interpolation functions h_i is that their values in natural coordinate system are unity at node, I, and zero at all other nodes of the element. The element displacements are also interpreted the same way as the geometry. For completeness, the functions are given as:

$$u = \sum_{i=1}^{8} h_i u_i\ , \quad v = \sum_{i=1}^{8} h_i v_i\ , \quad w = \sum_{i=1}^{8} h_i w_i$$

where u, v and w are displacements at any point in the element and u_i, v_i, w_i, I = 1 – 8 are corresponding nodal displacements in the coordinate system used to describe the geometry.

Three additional displacement "bubble" functions which have zero displacements at the surfaces are added in each direction for improved shear performance to form a 33 x 33 matrix. Static condensation is used to reduce this matrix to a 24 x 24 matrix at the corner joints.

Unlike members and shell (plate) elements, no properties are required for solid elements. However, the constants such as modulus of elasticity and Poisson's ratio are to be specified. Also, Density needs to be provided if self-weight is included in any load case.

Element stresses may be obtained at the center and at the joints of the solid element. The items that are printed are:

Normal Stresses: SXX, SYY and SZZ, where, say SXX represent the stress on the x-face of the solid element at a particular node, due to stress in x-direction.

Shear Stresses: SXY, SYZ and SZX, where, say SXY refers to the shear stress from torsion on both x- and y-faces of the solid element at a particular node, due to the shear stress on that particular node.

Principal stresses: S1, S2 and S3 are the stresses normal to the principal axes.

Von Misses stresses:

$$SIGE = \sqrt{(S1 - S2)^2 + (S2 - S3)^2 + (S3 - S1)^2}$$

Direction cosines: 6 direction cosines are printed, following the expression DC, corresponding to the first two principal stress directions. The flowchart for solid element finite element analysis using Staad.pro is shown in Fig. 3.

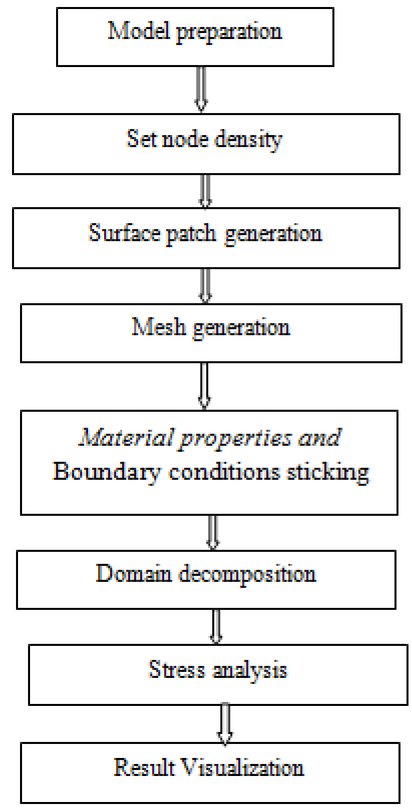

Fig. 3: Flowchart for Finite Element Analysis.

IV. DISCUSSION OF RESULTS

The finite element results generated from STAAD in the case of a pitched timber portal frame is presented. Various loads (e.g. dead load, imposed load, wind load etc) and load combinations in accordance to Eurocde5 provisions were applied to the frame as in the numerical example.

4.1 NUMERICAL EXAMPLE

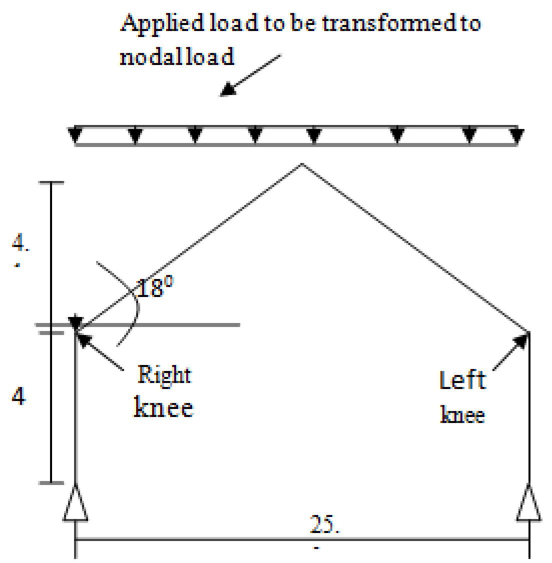

Fig. 4: Numerical Model

A 25m span portal frame shown in Fig. 4, with 4m column height to the eaves and 18^o roof pitch made from glued-laminated timber of strength grade GL24 was designed in accordance with the provisions of Eurocode5. The cross-sectional dimensions of the column and rafters are 100mm X 1200mm and 200mm X 1200mm respectively. The connection between the rafter and the column was assumed to be a rigid connection. A basic solid timber brick element of 300mm X 300mm and 100mm thick was used to model the entire timber frame. As shown in Fig. 5, the columns are comprised of single layer of the basic element in four rows.

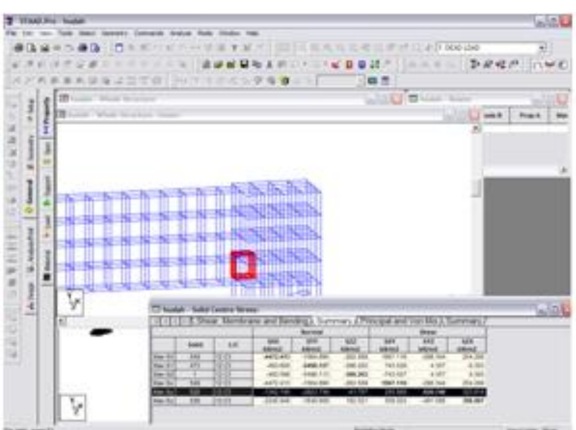

Fig. 5: Column/Rafter connection at the knee joint (with inset of result summary)

Similarly, the rafter has four rows but comprised of two layers. The column being two in number supports the rafter in-between, thereby creating a double shear connection in timber.

The fundamental commands for finite element analysis using STAAD require the definition of whether the problem is 2-Dimensional (2D) or 3-Dimensional (3D) (STAAD PLANE or SPACE). Next, defining the nodes, that forms the elements and their locations. Then the element incidences and their properties. Then, the definition of the supports conditions follows. Defining, the load cases and the load of each load case as nodal loads as well as the load combinations as defined in section 2.1. Lastly the analysis and the output format are defined.

Two methods are available for the creation of STAAD input files, i.e., the text editor and the Graphical User Interface (GUI). Whichever method is employed, the other is automatically generated. GUI was used to model the timber frame in this study because it is more users friendly. However, the text editor was also employed to capture the slanting (pitch) of the rafter because GUI would be more tasking.

As required for application of load on structures modelled with solid element, all the applied loads say, uniformly distributed, point load, etc other than the self-weight of the structure are transformed into nodal point loads. The text editor input command file for the analysis of the timber frame and a sample of the result are shown in appendix A.

By default the first joint coordinate is identified as 1 and is continued serially. The command Joint Coordinates is specified as a Cartesian coordinate system. Joints ere specified, using the global X, Y and Z coordinates. Also by default, the Element Incidence Solid command are generated from 1 and continued serially. Each element is identified by the element number followed the node numbers that formed the element. The Support command specifies the nodal positions and the respective types of supports prescribed at those nodes. In the framed structure analyzed pinned supports were prescribed.

The material command allows for the specification of the material characteristic, e.g. orthotropic, the density and Poisson ratio of the timber. The load types are placed at appropriate nodal points and the load combinations as allowed for in the code are prescribed. Finally, the analysis type, i.e. either static or dynamic is invoked.

Fig. 6.shows the finite element model of the timber frame analyzed using Staad.pro. It has 944 solid elements, joined with 1874 nodes. All the various load categories and combination were applied at the appropriate nodal points.

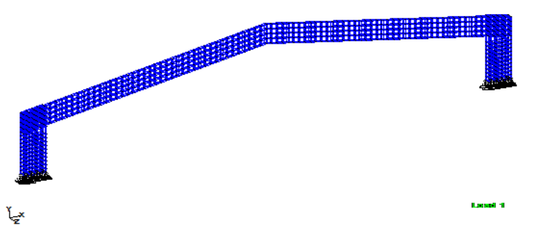

Fig. 6: Finite element model of a timber frame.

A single run of the program generates the stress/strain situation at the nodes of each element. It therefore makes it easy to study the equilibrium condition of the entire elements in the model including those at the joint area. The most critical values are also tabulated from which the one that is in the joint area is used in the design of the joints. Furthermore, once the element stress resultants have been computed for each node in the structure, the engineer must reduce this information to something useful for design. Since this study is limited to the design of the joints, generating contour plots of the shear stresses are useful because they provide an overview of the distribution of stresses making it easy to see not only the areas of high concentration, but also the inflection points where the stresses changes from positive to negative.

Since the bolts holding the columns and the rafters of frames together is in the Z-axis direction, the shear stress they resist is S_{ZX} which acts on the plane through which the bolts are passing. The critical shear stress in the joint area is approximately S_{ZX} =323 KN/m^2. The shear area is 300mm X 300mm. The equivalent shear load is therefore, 323 x 300 x 300 x 10^{-3} N = 29047N.

The contour page in the post processing mode of staad.pro (Staad.pro, 2003) allows one to view various stresses for solid elements, both graphically and numerically. The solid stress contours are color-based plot of the variation of stresses in a structure, given the load case and stress type the maximum and minimum values as well the distribution of stresses in the structure are shown. The contour type could be normal where the option uses the stress points at each corner and center of the solid elements or the enhanced color option that uses the same points as the normal contour plus the interpolated stress at the mid-point of the edges.

The second option takes more time to generate but it is more accurate. Plots and discussions of stress contours to illustrate the influence of load cases and load combinations considered in this research now follows. It clearly demonstrates the versatility of the finite element method algorithm employed in staad.pro [23].

The stress resultant contour plots highlighting the knee joints are shown in Figs. 7. to 14. The color code in the key to each of the S_{ZX} plots explains the distribution of stresses in the frame.

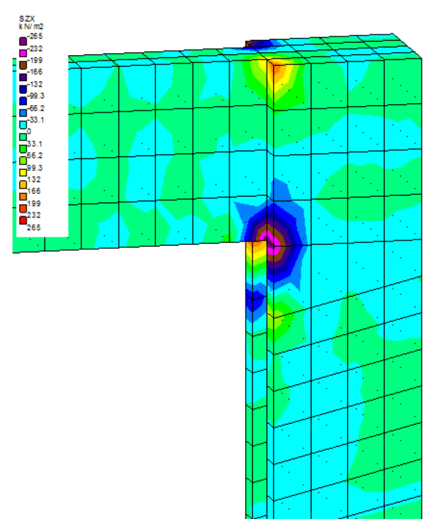

Fig. 7: Shear stress (S_{ZX}) contour plot showing the stress distribution on the right knee of the frame subjected to dead load.

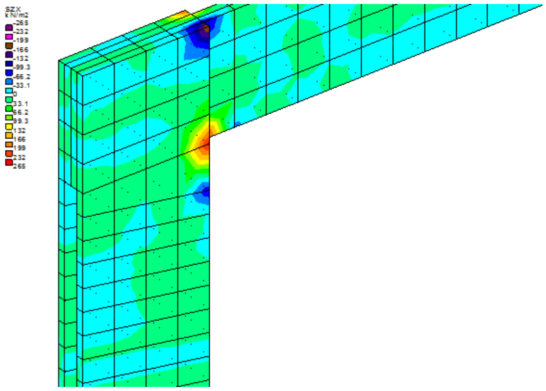

Fig. 8: Shear stress (S_{ZX}) contour plot showing the stress distribution on the left knee of the frame subjected to dead load.

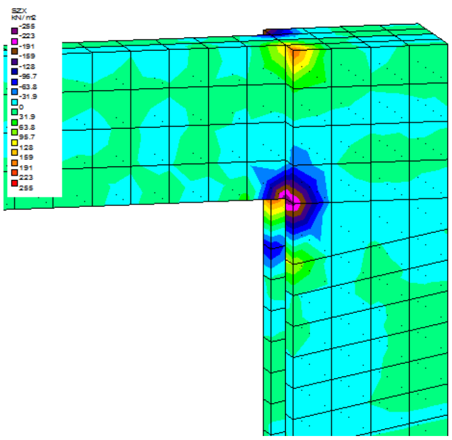

Fig. 9: Shear stress (S_{ZX}) contour plot showing the stress distribution on the right knee of the frame subjected to imposed load.

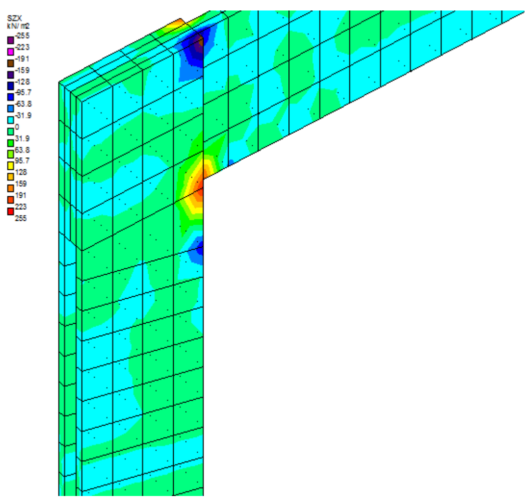

Fig. 10: Shear stress (S_{ZX}) contour plot showing the stress distribution on the left knee of the frame subjected to imposed load.

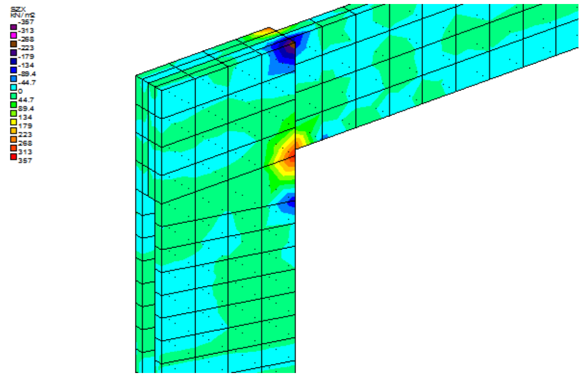

Fig. 11: Shear stress (S_{ZX}) contour plot showing the stress distribution on the left knee of the frame subjected to factored dead load (Load Combination C1).

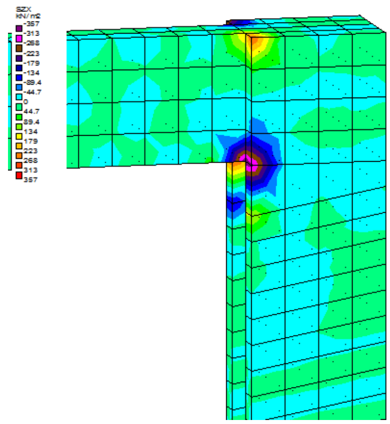

Fig. 12: Shear stress (S_{ZX}) contour plot showing the stress distribution on the right knee of the frame subjected to factored dead load (Load Combination C1).

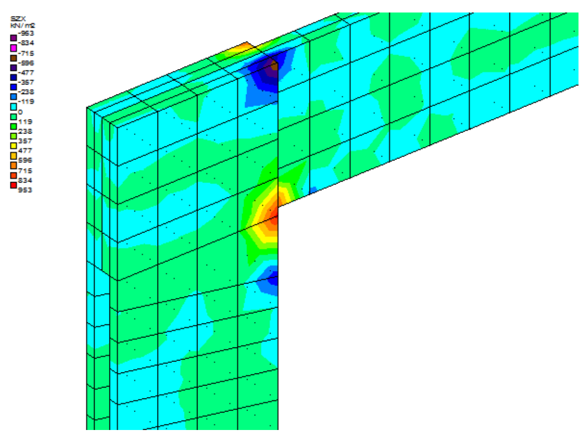

Fig. 13: Shear stress (S_{ZX}) contour plot showing the stress distribution on the left knee of the frame subjected to factored dead and factored full snow load (Load Combination C3).

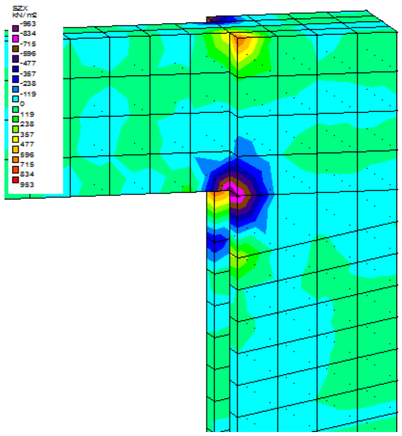

Fig. 14: Shear Stress (S_{ZX}) contour plot showing the stress distribution on the right knee of the frame subjected to factored dead and factored full snow load (Load Combination C3).

Figs. 7 and 8 for the right and left knee joint respectively, of the frame subjected to dead load only, shows the non-uniformity of the shear stress (S_{ZX}) distribution. The stresses for the entire structure varied between -265 KN/m^{2}and 265 KN/m^2. However, the color code shows the shear stress S_{ZX} around most of the joint area lies between -99.3 KN/m^2 and 99.3 KN/m^2. At the corners between the rafters and columns of the joints, concentrated stress as high as -199 KN/m^2 to 199 KN/m^{2}are observed. This however is outside the location of the fasteners since the code stipulates edge distances. Hence the high stress does not affect the joint mechanism.

The shear stress distribution pattern is similar for the imposed load, Figs. 9 and 10 for right and left knee respectively. It ranges from -255 KN/m^2 to 255 KN/m^2 for the whole structure. The full snow load (Figs. 9 and 10) however, has a higher range of stress distribution (i.e. ranges from -397 KN/m^2 to 397 KN/m^2) for the entire elements of the frame. Stresses around the joint area are between -199 KN/m^2 and 199 KN/m^2. The concentrated shear stress as high as -298 KN/m^2 to 298 KN/m^{2}are observed at corners of the joint.

As for the factored loads, that is the load combinations C1 to C5, the factored dead load C1 (in Figs. 11 and 12) gave shear stress range of -357 KN/m^{2}to 357 KN/m^2 for the entire frame structure. The distribution of the stresses in the joint area is between -139 KN/m^2 and 139 KN/m^2 and a concentrated -288 KN/m^2 to 288 KN/m^2 at the corners of the joint.

Combination C3 in Figs. 13 and 14, for factored dead and factored full snow loads generated shear stresses distributions of -963 KN/m^2 to 963 KN/m^2. The stresses in the joint area are -357 KN/m^2 to 357 KN/m^2.

Therefore, the shear stress (S_{ZX}) distribution contours showed that load combination C3 for factored dead and factored full snow loads is the most critical value to be used for the design of joints in timber frames. However, combination C2 for the factored dead and factored imposed loads is also significant as it is only about 23% less than the stress value for combination C3 and also it is peculiar to local environment.

CONCLUSION

Displacement-based finite element method was used to realistically as possible, model the frame structure using solid elements to present the geometry. All the applicable loadings and their appropriate combinations were applied and responses were obtained in a form that allows for the design of timber joints to Eurocode5 provisions. The stress distribution in the joint area is not uniform and ranged between compressive and tensile conditions. Combination of factored dead and factored full snow load gave most critical load action that determined the design of the joint. However, the combination of factored dead and factored imposed load also gave appreciable resultant joint load.

REFERENCES

[1] Morais, J. L. Xavier, J. C. Dourado, N. M. and Lousada, J. L. (1999); "Mechanical behaviour of wood in the orthotropic directions". Department of Engineering, ICET/UTAD, Portugal.

[2] Chamis, C. C. and Brinson, J. H. (1977); "Ten-deg off-axis test for shear properties in fibre composites". Exp. Mech., September: 339-346

[3] Voloshin, A. and Arcan, M. (1980); "Failure of unidirectional fibre-reinforced materials – new

methodology and results". Exp. Mech., August: 280-284

[4] Walrath, D. E. and Adams, D. F. (1983); "The Iosipesco shear test as applied to composite materials". Exp. Mech. August: 280-284

[5] Sims, G. D. (1992); "Development in harmonisation of standards for polymer matrix composites". Composites Testing and Standardisation, Hoggs, P. J. et al. (cds.), ECMM-CTS, EACM, September: 3-20

[6] Hojo, M., Sawada, Y. and Miyairi, H. (1994); "Influence of clamping method on tensile properties of unidirectional CFRP in $0^o$and 90^o directions – round robin activity for international standardisation in Japan". Composites, 25 (8): 786-796

[7] Hung, S. C. and Liechti, K. M. (1999); "Finite element analysis of Arcan specimen for fibre reinforced composites under pure shear and biaxial loading".J. Comp. Mat., 33 (14): 1288-1317

[8] Odegard, G. and Kumosa, M. (1999); "Elastic-plastic analysis of Iosipescushear test".J. Comp. Mat. 33 (21): 1981-2001.

[9] Liu, J. Y. (1984); "New shear strength test for solid wood". Wood and Fibre Science, 16 (4): 567-574

[10] Silver, A., and Yu, Y. (1993); "Elastic constants for hardwood measured from plate and tension tests". Wood and Fibre Science, 25 (1): 8-12

[11] Silver, A., Ying, Y., Timothy, W. and Wenjie, Z. (1994);"Orthotropic elastic constants for hardwood measured from plate and tension tests". Wood and Fibre Science, 26 (1): 107-121

[12] Oudjehane, A. and Raclin, J. (1995);"On the influence of orientation upon the mechanical behaviour of oak wood in a general state of stress". Wood Sci. Techn., 29 (1): 1-10

[13] Liu, J. Y., Ross, R. J. and Rammer, S. R. (1996);"Improved Arcan shear test for wood". Proc. of the International Wood Engineering Conference, Gopu and Vijaya, K. A. (cds.), New Orleans, 2, 85-90

[14] Liu, J. Y. (2000); "Effect of shear coupling on shear properties of wood". Wood and Fibre Science, 32 (4); 458-465

[15] Cheung, C. K. and Sorenson, H. C. (1983); "Effect of axial loads on radial stress in curved beams". Wood and Fibre Science, 15 (3): 263-275

[16] Zalpeh, B. L. and McLain, T. E. (1992);"Strength of wood beams with fillet interior notches: a new model". Wood and Fibre Sciences, 24 (2): 204-215

[17] Pellicane, P. J. and France, N. (1994**);**"Modelling wood pole failure, Part 1: finite element stress analysis". Wood Sci. Techn., 28 (3): 219-228

[18] Bouchair, A and Vergne, A. (1995);"An application of Tsai criterion as a plastic flow law for Timber bolted modelling". Wood Sci. Techn., 30 (1): 3-19

[19] C.E.B. (1980); "Structural safety". Bulletin d' information N°127 and 128, Brussels, Belgium.

[20] CEN-TC250, (2004);"EN 1995-1-1 Design of timber structures, Part 1-1.. General rules and rules for buildings". Brussels.

[21] Racher, P. and Rouger, F (1994); "Serviceability limit states-A proposal for updating Eurocode 5 with respect to Eurocode 1".In: CIB W18A-27th Meeting, Sydney, Australia, July 1994.

[22] Staad.pro (2003); "Research Engineers International". NetGuru Incorporation. USA

[23] Melosh, R. J. (1963); "Basis for Derivation of Matrices for the Direct Stiffness Method". AIAA journal, Vol. 1, No. 7, pp 1631-1637