Composite Materials in the Fasteners of glued wooden Structures

G G Kashevarova and M A Vodiannikov
Perm National Research Polytechnic University, 29, Komsomolsky Prospect, Perm, 614990 Russia

ggkash@mail.ru

Abstract. This paper presents the data obtained from experiments and numerical simulation of contacts in ANSYS program by the example of a rigid joint of glulam beam using composite connection elements. The tests results of a continuous beam and a beam with a joint using composite dowel pins and joint plates are shown. ANSYS methods analysis of contacts between different building materials with consideration of their nonlinear interaction is described. The fields displacements and areas of contact interactions are shown. Conclusions of simulation and use of composite materials in ANSYS program are drawn. The comparison of the numerical results and full-scale tests was made.

1. Introduction
Design and simulation of glulam rigid joints structures is the primary task during the construction of long-span industrial, civil buildings and facilities. Solving this task allows to eliminate some problems connected to manufacturing and transportation of big prefabricated glulam structural components. The design of full-strength joints is important during the construction of steel and reinforced concrete structures respectively. However, this task appears to be more complicated for timber and glulam structures because wood is a natural building material which can not be obtained by any means, than growing it in natural environment.

One of the ways to solve the task is by applying modern methods of computer simulation using software which considers non-linear and orthotropic characteristics of materials during the contact analysis. Simulation using special software allows making a precise description of composite material work, timber in particular. Numerical simulation enables to compare different variants of structural design, make quantity and quality estimation of analysis methods and obtain the data on the work of the structure inside the components.

This article provides the estimation of the glulam beam bearing capacity with rigid joint in the middle of the span performed using inserted diagonal dowel pins made of composite material, i.e. carbon fiber reinforced plastic. The type of failure was defined considering contact interaction. The analysis is performed in ANSYS software (ANSYS Customer number 1062978) [1].

2. Experiment simulation
In order to perform strain-stress state analysis, finite-element model of a glulam beam with a joint in the middle of its span was created. The model is performed in AutoCAD program (Figure 1) and exported into ANSYS program. The calculation of the bended beam was performed (refer to Figures 1
Numerical and full-scale tests were performed for two types of the elements: continuous glulam beam and simply supported beam with a glued-in dowel joint.

The material used for the glulam timber beam is second quality wood (pine) [2, 3]. The analysis model is discretized with finite elements SOLID185 which are used for simulation of 3D solid objects. It includes 8 nodes each of which has three degrees of freedom. Finite element SOLID185 may have such properties as yield ability and plastic flow and other assigned parameters.

Carbon fiber dowel pins [4] which are glued into the timber are presented by dowels with 5 mm diameter. Finite element assigned for dowel pins is BEAM188, it has six degrees of freedom in each node. Joint plates on the top and bottom of the beam which help to assemble the whole structure.

Figure 1. Experiment simulation.

Figure 2. Overall view of the testing facility with the attached sample.
together were presented as bidirectional carbon fiber reinforced plastic with synthetic polymeric matrix. Finite element SHELL281 is assigned for carbon fiber joint plates. This finite element is used for analysis of plates with assigned characteristics; it has 8 specifying nodes with 6 degrees of freedom. It is considered to be the most reasonable for the simulation of layered composite structures.

Glued wood to wood and carbon fiber to wood connections in the analysis were considered with non-linear properties due to friction forces in contact area. Thus, special contact elements are introduced into the model.

In such tasks, one of the surfaces is conditionally called a contact surface, and the other one is accepted as a target surface, they are simulated with finite elements CONTA174 and TARGE170 accordingly [6]. Finite elements which compose a contact pair have a common set of properties.

The assignment of the material properties is performed by the parameterization of their physical properties. The parameterized properties of wood [7] and carbon fiber [8] are shown in Tables 1, 2.

**Table 1.** Wood properties.

**Table 2.** Carbon fiber properties.
3. Analysis of results
During the numerical simulation in ANSYS program, the nonlinear analysis was carried out by incremental loading increasing by 1.5 kN. The following results were obtained. The failure of both samples is due to formation of a plastic hinge in the beam body under the movable pivot point in the zone of maximum deflection moment.

Plastic deformations of the sample with rigid joint occur upon applied load 10.5 kN, its failure occurs upon the load 19.5 kN. The fields of displacements and types of contact are shown in Figures 3 and 4. The maximum forces in composite components amount 697 MPa. The fields of stress distribution in the dowels and plates are shown in Figure 5.

![Figure 3. Fields of displacements.](image1)

![Figure 4. Fields of contact types.](image2)

![Figure 5. Fields of stresses in composite dowel sand joint plates.](image3)

4. Experiment results
Sample test was carried out in accordance with analysis diagram on a specially designed testing bench (Figure 2). The tests were conducted until sample failure by applying incremental load. The load on the beam’s free end was applied using a hydraulic jack.

During the test of the continuous beam, the following testing parameters were considered: the shift of the beam’s free end and the force value registered by the jack pressure gauge. During the test of the joint with carbon fiber components, the width of joint gap at both sides of the beam was also considered.
Joint plates were manufactured of bidirectional carbon fiber plates with 5 mm width, the dowels were manufactured of carbon fiber rodding with 5 mm diameter. The dowel pins were glued in with synthetic epoxy adhesive.

Actual plastic deformations of the sample with rigid joint occur upon the applied load 11.5 kN, its failure is registered when the loading is up to 24.4 kN. The maximum width of joint gap at the bottom is registered at average loading value 3.7 mm, whereas the initial joint gap was equal to 1.5 mm (Figure 6).

Plastic deformations in the continuous sample occur when the load is 11.2 kN, its failure (when the same load is applied but shifts increase) occurs when the load is 19.2 kN.

The diagrams of the displacement’s dependence of the beam’s free end on the load are shown in Figure 7. It presents experimental data for a continuous beam and a beam with a joint carbon fiber component, as well as ANSYS analysis results for a joint with carbon fiber component.

The failure of both beams results from plastic hinge under the free support in the middle of the span. Furthermore, in the moment of failure, up to 5 mm wide cracking was observed in the beam body in the neutral zone of the joint. This signifies that shear stresses appear due to mechanical impact of the rodding on the wood layers. Some difference in design and actual forces and shifts is provoked by wood flaws, as well as the use of unified design parameters $\sigma_B$ as strength properties set in regulatory documents. It should be noted, that both beams returned into their initial state after load removal.
The bearing capacity of the sample with rigid joint is higher because the inserted dowels act as reinforcing elements preventing the reciprocal displacement of timber layers, which diminishes layer deformation of the structure. The forces in joint plates, rods and glued connections do not exceed critical values [9].

5. Conclusions
Numerical simulation of composite materials (glulam and carbon fiber) in ANSYS program enables to perform the strain-stress state analysis of complex structures and their joints with consideration of multiple factors. One of the significant factors is contact interaction. In addition, the model considers such aspects as suddenly occurring and ceasing friction forces.

Experimental confirmation of the data obtained from the numerical analysis proves the correct choice of the model. The values of shifts depending on load in numerical and full-scale tests verify each other. The spread in values is caused by wood structure imperfections (flaws), as well as the application of standardized material characteristics instead of their actual characteristics. The estimation of the forces dependence final diagrams on structural shifts proves the correctness of the chosen analysis model and possibility of its use to design real structures.

References
[1] Basov K A 2002 ANSYS Examples (Moscow: ComputerPress)
[2] SP 64.13330.2011 2011 National construction regulations “Wood constructions” (Moscow: JST “CPP”)
[3] Kalugin A 2008 Wooden Constructions: University Tutorial (Moscow: ASV Press)
[4] Gappoev M 2004 Wooden and Plastic Constructions (Moscow: Association of construction universities)
[5] SP 64.13330.2011 2011 National construction regulations “Wood constructions” (Moscow: JST “CPP”)
[6] Vodiannikov M A 2017 Analysis of wood structure connections using cylindrical steel and carbon fiber dowel pins IOP Conf. Ser.: Mater. Sci. Eng. 205 012031
[7] Lennartz M W 2016 New Architecture in Wood: Forms and Structures (Basel: Birkhauser Verlag GmbH) 1
[8] Gugutsidze G and Draskovic F 2010 Slovak J. Civil Eng. 2 1–6
[9] Reichmanis E 2003 High Performance Carbon Fibers American Chemical Society Journal 1 2–4