

Advanced Finite Element Modeling of Wood Structures: A Technical Guide to Abaqus Implementation and Rheological Analysis

Published: April 20, 2025 | Index ID: #120

Wood is one of the oldest and most versatile construction materials, yet its inherent biological nature introduces a level of complexity that challenges traditional structural engineering methods. Unlike steel or concrete, wood is a naturally grown, cellular material characterized by orthotropic properties, hygroscopic sensitivity, and complex time-dependent behaviors. To accurately predict the long-term performance of timber structures—particularly under varying environmental conditions and mechanical loads—advanced numerical simulations are required. Among the most powerful tools for this task is **ABAQUS**, a high-end Finite Element Analysis (FEA) suite capable of handling the nonlinearities associated with timber rheology.

The Theoretical Framework of Wood as a Material

Before implementing a numerical model in ABAQUS, one must understand the fundamental physical characteristics that define wood. Engineering wood design is grounded in the recognition of its **anisotropic nature**. Specifically, wood is **orthotropic**, meaning its mechanical properties are unique and independent in three mutually perpendicular directions: Longitudinal (L), Radial (R), and Tangential (T). The longitudinal axis corresponds to the direction of the grain (fiber), while the radial and tangential axes represent the growth rings' orientations.

Orthotropic Elasticity and the Stiffness Matrix

In a linear elastic framework, wood requires nine independent elastic constants to define its behavior: three moduli of elasticity (EL, ER, ET), three shear moduli (GLR, GLT, GRT), and three Poisson's ratios (ν_{LT} , ν_{RT} , ν_{RL}). The relationship between stress (σ) and strain (ϵ) is expressed through the compliance matrix. In ABAQUS, this is typically defined within the ***ELASTIC, TYPE=ENGINEERING CONSTANTS** option. However, wood rarely remains in a linear elastic state under long-term loading or high stress concentrations, such as those found in dowel-type connections.

Hygroscopicity and Mechano-Sorption

Perhaps the most significant challenge in wood modeling is its **hygroscopic nature**. Wood's moisture content (MC) significantly impacts its mechanical strength and stiffness. Furthermore, wood exhibits a phenomenon known as **mechano-sorptive deformation**. This is an additional deformation that occurs when wood is subjected to simultaneous mechanical load and moisture content variation. Traditional creep models (which are time-dependent) cannot fully capture this behavior, necessitating a rheological model that incorporates both time and moisture as variables.

The Mirianon-Fortino-Toratti Methodology

In the seminal 2008 report, *"A Method to Model Wood by Using ABAQUS Finite Element Software,"* researchers Florian Mirianon, S. Fortino, and T. Toratti established a comprehensive framework for structural analysis focused on the long-term response of wood. This methodology is particularly notable for its application to **dowel-type connections**, which are critical failure points in timber engineering.

System Requirements and Infrastructure

Executing the Mirianon method requires more than just the base ABAQUS installation. Because standard ABAQUS material libraries do not contain a built-in wood constitutive model that accounts for mechano-sorption and complex rheology, users must leverage **User Subroutines (UMAT)**. The following prerequisites are mandatory:

- ABAQUS CAE/Solver: For geometry creation, meshing, and solving.
- Fortran Compiler: Required to compile the UMAT or VUMAT subroutines. The compiler must be strictly compatible with the specific version of ABAQUS in use (e.g., Intel Fortran).
- Effective Computing Hardware: High-memory (RAM) and multi-core processors are essential, as nonlinear wood models involving moisture diffusion are computationally expensive.

Technical Breakdown: The Rheological Model for Wood

The core of the Mirianon method involves a **rheological model** that partitions total strain into several components. Total strain (ϵ_{total}) is the sum of:

- ϵ_{e} : Instantaneous elastic strain.
- ϵ_{v} : Viscoelastic creep (time-dependent).
- ϵ_{ms} : Mechano-sorptive strain (moisture-variation dependent).
- ϵ_{sw} : Swelling and shrinkage (directly related to changes in MC).
- ϵ_{p} : Plastic strain (irreversible deformation).

Implementation via UMAT Subroutine

The **UMAT (User Material)** subroutine in ABAQUS allows engineers to define the material Jacobian (the derivative of the stress increment with respect to the strain increment) and update the stress state at each integration point. For wood, the UMAT must track the internal state variables (ISVs) representing the historical loading and moisture exposure. This is critical for predicting the "memory effect" of wood fibers under cyclic humidity changes.

Mathematical Representation of Creep

Creep in wood is often modeled using a series of **Kelvin-Voigt elements**. Each element consists of a spring and a dashpot in parallel. The total viscoelastic strain is the summation of the strains in these elements. In the ABAQUS environment, this requires the solver to integrate the differential equations governing the dashpot behavior over the time increment (Δt).

Step-by-Step Procedure for Wood Modeling in ABAQUS

Achieving a close-to-reality simulation involves a systematic workflow. Below is a structured guide for setting up a wood model based on the VTT Technical Research Centre of Finland's standards.

1. Defining the Local Coordinate System

Because wood is orthotropic, the global coordinate system (X, Y, Z) rarely aligns with the grain direction. In ABAQUS, users must define a ***ORIENTATION** assignment to align the L, R, and T directions with the geometry of the timber member (e.g., along the longitudinal axis of a beam).

2. Constitutive Material Definition

Users must input the density and the orthotropic elastic properties. If using the UMAT for long-term behavior, the properties are passed into the subroutine. The density of wood is a function of its moisture content, which must be accounted for if mass-dependent analyses (like vibration or sound synthesis) are performed.

3. Modeling the Loading and Environmental Steps

Simulating wood behavior often requires multiple steps:

- 1. Step 1 (Equilibrium): Establish initial moisture content and temperature.
- 2. Step 2 (Loading): Apply mechanical loads (e.g., tension on a dowel).
- 3. Step 3 (Environmental Cycling): Introduce variations in relative humidity (RH) to trigger mechano-sorptive effects. This is done using the *Predefined Field or a coupled temperature-displacement analysis where "temperature" acts as a proxy for moisture.

Comparative Analysis: Wood Modeling Strategies

Depending on the goal of the analysis, different levels of complexity may be chosen. The table below compares the standard approach versus the advanced rheological approach.

Feature	Linear Orthotropic Model	Advanced Rheological Model (Mirianon)
Complexity	Low - Standard ABAQUS Input	High - Requires UMAT & Fortran
Time Sensitivity	None (Instantaneous)	High (Creep & Relaxation)
Moisture Impact	Ignored or Static	Dynamic (Mechano-sorptive)
Failure Mode	Simplified (Stress-based)	Detailed (Ductile/Brittle Analysis)
Application	Short-term structural checks	Serviceability Limit State (SLS) analysis

Case Study: Dowel-Type Connection Performance

One of the primary applications of the ABAQUS wood model is the analysis of **dowel-type connections**. These connections involve steel bolts or dowels inserted into pre-drilled holes in timber. The interaction between the steel (isotropic/elasto-plastic) and the wood (orthotropic/visco-plastic) creates complex stress states.

Contact Mechanics and Friction

In ABAQUS, the contact between the dowel and the wood hole must be defined using ***CONTACT PAIR** or ***GENERAL CONTACT**. A friction coefficient (typically 0.3 to 0.7 for steel-on-wood) is applied. The simulation must account for the **crushing of wood fibers**(embedment strength). By using a nonlinear material model, the simulation can show the localized densification of wood fibers under the dowel, which provides a more accurate load-carrying capacity than simple linear models.

Numerical Instabilities and Convergence

Modeling wood failure often leads to numerical convergence issues due to the softening behavior of wood in compression and its brittle nature in tension. Techniques such as **automatic stabilization** (using the *STATIC, STABILIZE option) or switching to **ABAQUS/Explicit**for dynamic failure events can help manage these instabilities. However, for long-term creep, the Standard (Implicit) solver remains the industry preference.

Practical Implementation Checklist for Engineers

To ensure a robust simulation, the following checklist should be followed by technical writers and lead engineers:

- **Verify Fiber Orientation:** Ensure the longitudinal direction (L) is correctly mapped to the grain. Misalignment by even 5 degrees can result in a 30% error in stiffness.
- **Moisture Step Calibration:** If modeling swelling, ensure the swelling coefficients (ϵ_{sw} ; ϵ_{sw}^T ; ϵ_{sw}^R) are attributed to the specific wood species (e.g., Norway Spruce vs. Douglas Fir).
- **Element Selection:** Use C3D8R (8-node linear brick, reduced integration) for most structural timber models to avoid shear locking, but ensure a fine mesh in contact zones.
- **Subroutine Verification:** Always run a single-element test (the "patch test") with the UMAT to verify that the stress-strain output matches analytical solutions for simple creep.

Summary and Broader Implications for Timber Engineering

The ability to model wood with high fidelity in ABAQUS marks a significant advancement in timber engineering. As

the construction industry moves toward sustainable, massive timber structures like Cross-Laminated Timber (CLT) and Glulam skyscrapers, the importance of predicting long-term structural integrity becomes paramount. The methodologies developed by Mirianon and colleagues provide the necessary framework to bridge the gap between empirical testing and predictive engineering.

By integrating rheological models with finite element software, engineers can now account for the "living" nature of wood—its response to the environment, its gradual deformation over decades, and its complex failure mechanisms. This not only leads to safer structures but also allows for more efficient use of wood material, reducing over-design and promoting the use of timber in increasingly ambitious architectural projects. The future of wood modeling lies in the refinement of these constitutive laws and the democratization of user subroutines, making advanced FEA accessible to a wider range of structural designers.

Tags: #Abaqus #Finite Element Analysis #Wood Modeling #Timber Engineering #Rheology #UMAT

